Introduction to PolyUMod & MCalibration

Veryst Engineering, LLC.

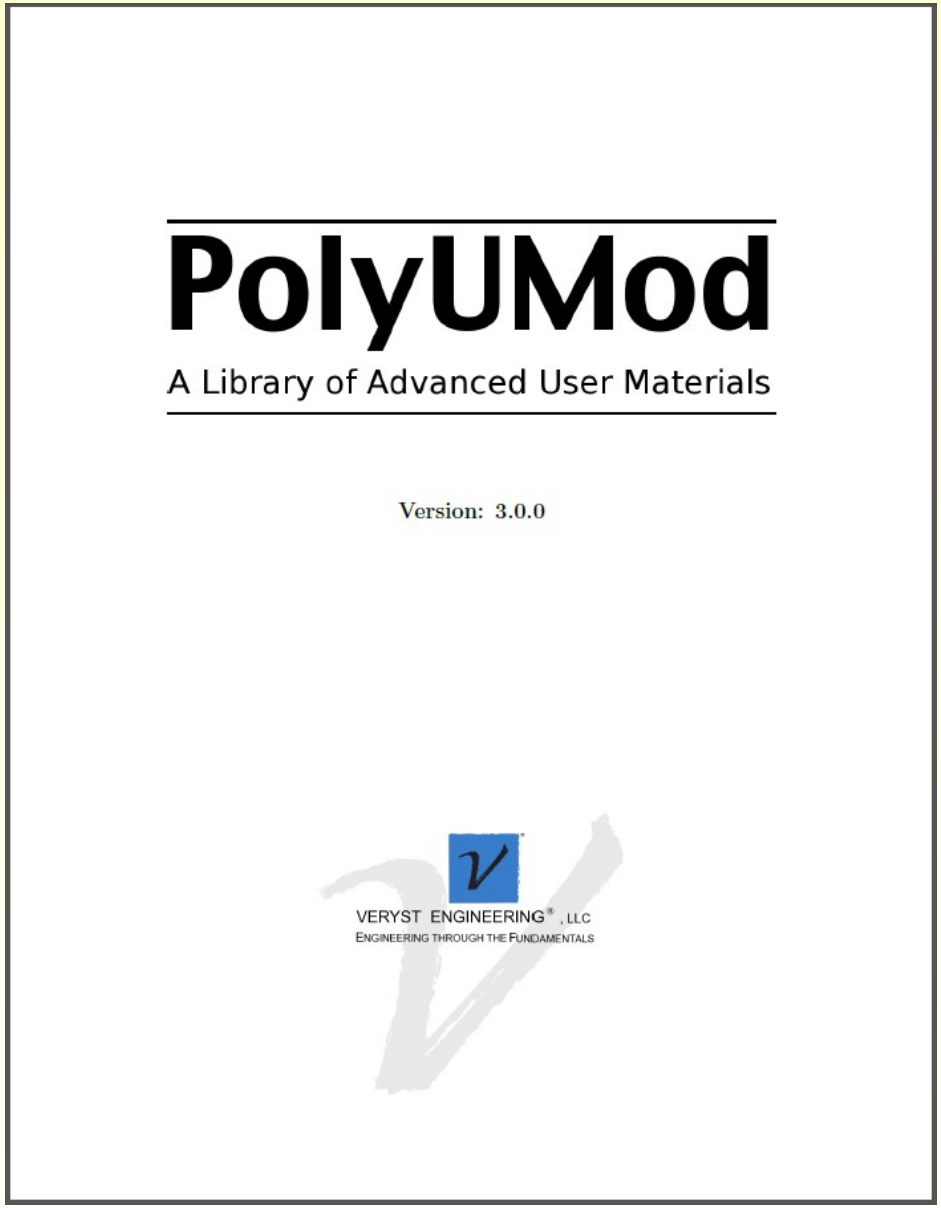
(781) 433 - 0433

Needham, MA

Copyright Veryst Engineering , 2015

Outline

- Introduction
- Installation and Files
- PolyUMod Models
- MCalibration Workflow



Copyright Veryst Engineering , 2015

Introduction - PolyUMod®

- **PolyUMod** is a library of advanced user material models for Abaqus, ANSYS, and LS-DYNA.
- **PolyUMod** is delivered as a shared library file (dll on Windows, and so on Linux).
- **PolyUMod** is a FE software plugin which is used when calculating the stresses for a given strain increment.
- The **PolyUMod** actions cannot be directly seen. The software is controlled by the provided global and material-specific parameters.

Copyright Veryst Engineering , 2015

Introduction - MCalibration®

- **MCalibration** is a GUI application for Windows and Linux computers.
- **MCalibration** can be used to calibrate all material models in the PolyUMod library, and all native material models in Abaqus and ANSYS.
- **MCalibration** takes existing experimental data and then finds the best material parameters for the selected material model.
- The calibrated material model can be exported to Abaqus/CAE and ANSYS WB.

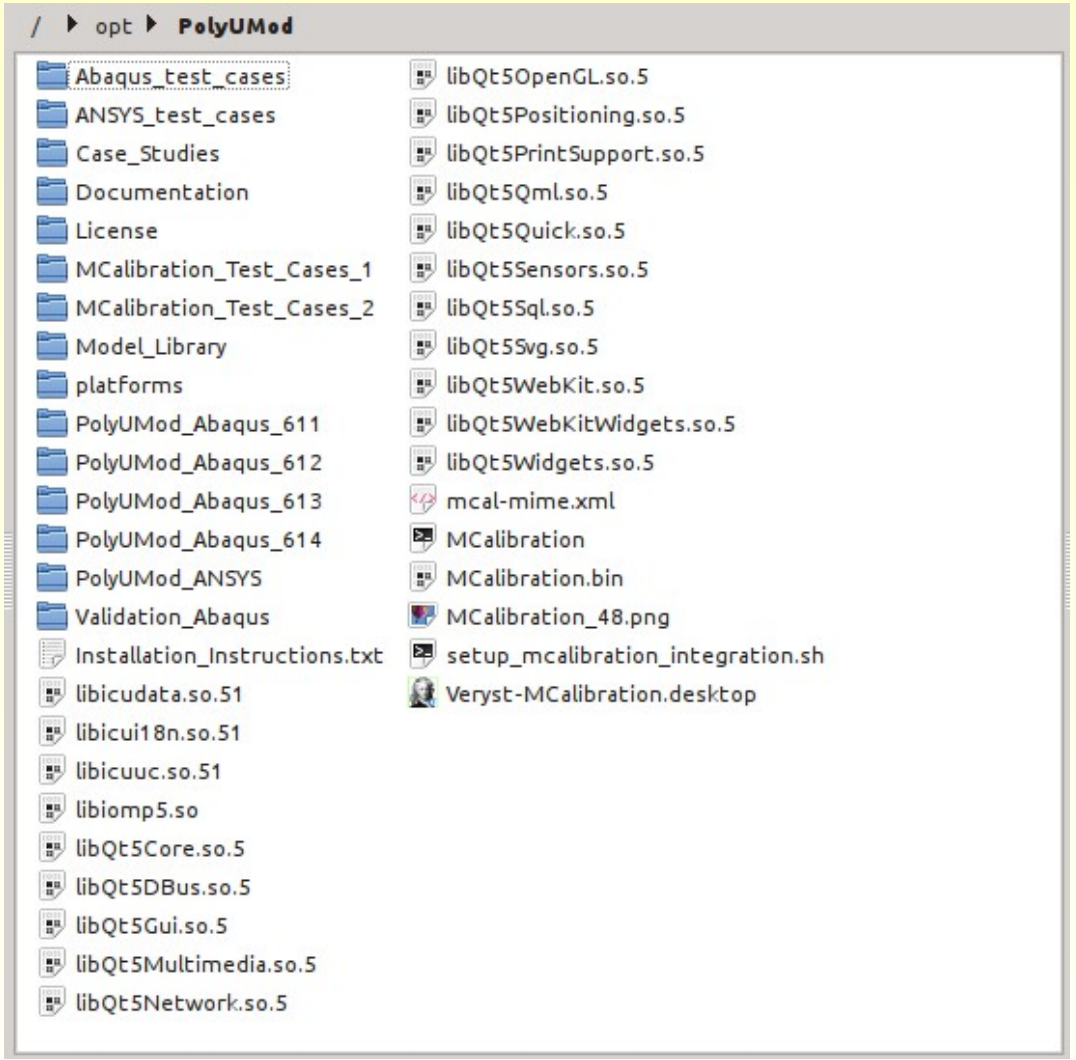
Copyright Veryst Engineering , 2015

Installation & Files

The PolyUMod and MCalibration installer puts all files in:

/opt/PolyUMod
(for Linux)

C:\Program Files\PolyUMod
for (Windows)



Copyright Veryst Engineering , 2015

License Server

- The PolyUMod and MCalibration software is controlled by the RLM network license server http://www.reprisesoftware.com/RLM_Enduser.html
- To run the PolyUMod and MCalibration software it is necessary to have a valid license file and a running license server.
- The license file is typically provided as part of a yearly license agreement.
- The license server can be run on any Windows or Linux computer on the local network.

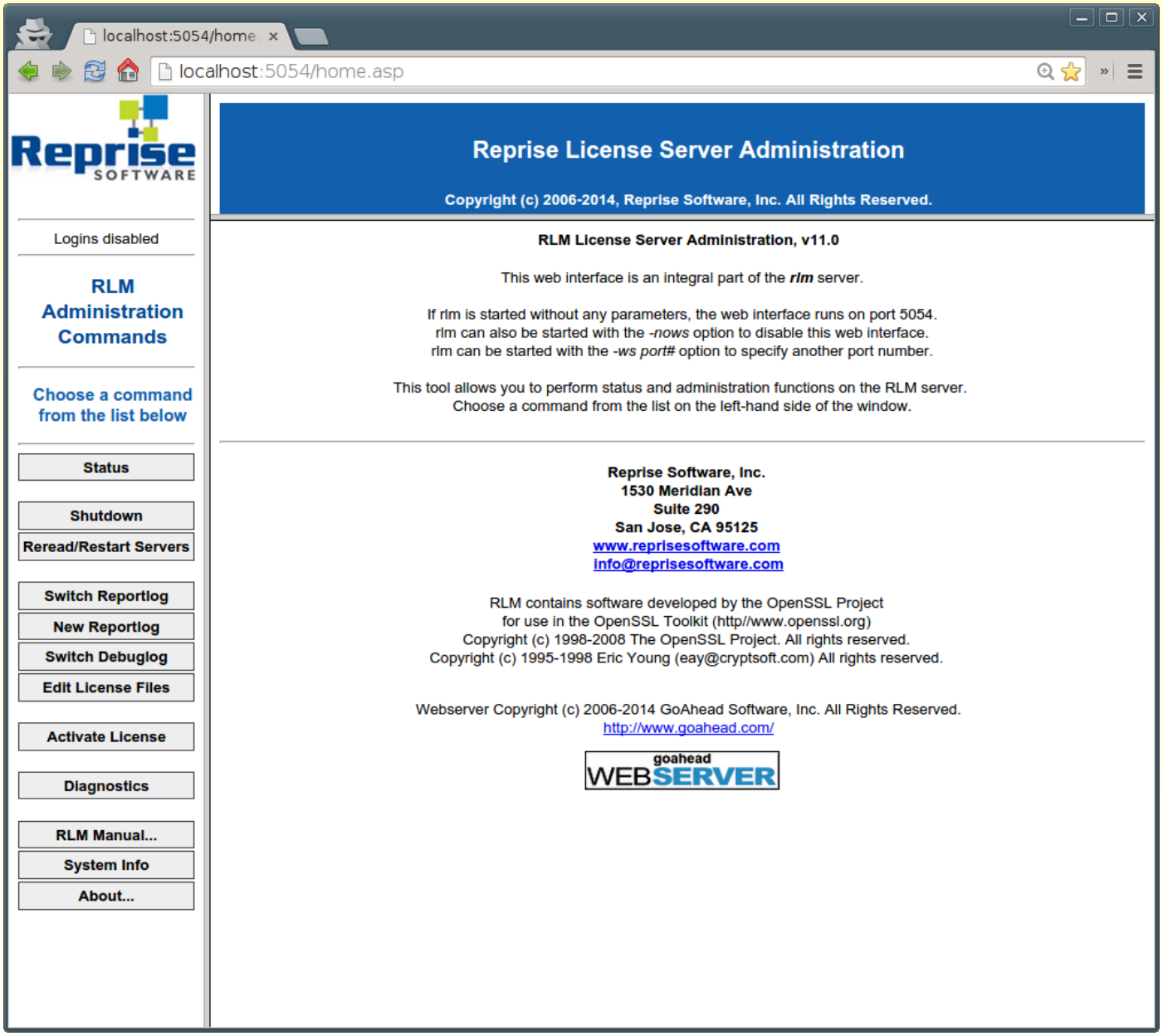
Copyright Veryst Engineering , 2015

License Server

The license server can be controlled through command line arguments, or using a web interface at:

http://localhost:5054

where localhost is the name of the license server.


localhost:5054/home
localhost:5054/home.asp
Reprise SOFTWARE
Logins disabled
RLM Administration Commands
Choose a command from the list below
Status
Shutdown
Reread/Restart Servers
Switch Reportlog
New Reportlog
Switch Debuglog
Edit License Files
Activate License
Diagnostics
RLM Manual...
System Info
About...
Reprise License Server Administration
Copyright (c) 2006-2014, Reprise Software, Inc. All Rights Reserved.
RLM License Server Administration, v11.0
This web interface is an integral part of the rlm server.
If rlm is started without any parameters, the web interface runs on port 5054.
rlm can also be started with the -nows option to disable this web interface.
rlm can be started with the -ws port# option to specify another port number.
This tool allows you to perform status and administration functions on the RLM server.
Choose a command from the list on the left-hand side of the window.
Reprise Software, Inc.
1530 Meridian Ave
Suite 290
San Jose, CA 95125
www.reprisesoftware.com
info@reprisesoftware.com
RLM contains software developed by the OpenSSL Project
for use in the OpenSSL Toolkit (http//www.openssl.org)
Copyright (c) 1998-2008 The OpenSSL Project. All rights reserved.
Copyright (c) 1995-1998 Eric Young (eay@cryptsoft.com) All rights reserved.
Webserver Copyright (c) 2006-2014 GoAhead Software, Inc. All Rights Reserved.
http://www.goahead.com/
goahead WEBSERVER

Copyright Veryst Engineering , 2015

License Server

- One easy way to check if the software has been installed properly is to try to start MCalibration. If it starts without problems then the license server works.
- Run the test cases in:
  /opt/PolyUMod/Abaqus_test_cases
  /opt/PolyUMod/ANSYS_test_cases

  or
  C:\Program Files\PolyUMod\Abaqus_test_cases
  C:\Program Files\PolyUMod\ANSYS_test_cases
  to test if the PolyUMod library has been installed properly.

Copyright Veryst Engineering , 2015

PolyUMod Material Models

1. Linear elastic (LE)
2. Neo-Hookean (NH)
3. Eight-chain (EC)
4. Bergstrom-Boyce (BB)
5. Bergstrom-Boyce with Ogden-Roxburgh Mullins effect (BBM)
6. Anisotropic Bergstrom-Boyce with Ogden-Roxburgh Mullins effect (ABBM)
7. Hybrid model (HM)
8. Multinetwork model (M8)
9. Arruda-Boyce viscoplasticity model (AB)
10. Dual Network Fluoropolymer (DNF) model
11. Three Network Model (TNM)
12. Anisotropic eight-chain model (AEC)
13. Micromechanical Foam Model (MFM)
14. Parallel network model (PNM)
15. Three Network Foam Model (TNFM)
16. Dynamic Bergstrom-Boyce model (DBB)
17. Silberstein-Boyce-1 (SB1)
18. Silberstein-Boyce-2 (SB2)
19. Flow Element Networks (FEN) model
99. Multi-Temperature model

The most commonly used models are marked in blue

Copyright Veryst Engineering , 2015

PolyUMod Material Parameters

Line 1:

1	2	3	4	5	6	7	8
MM	ODE	JAC	ERRM	TWOD_S	VERB	VTIME	VELEM

Line 2:

9	10	11	12	13	14	15	16
VINT	ORIENT	NPROP	NHIST	MU	KAPPA	FAILT	FAILV

- All PolyUMod material models use an initial 16 global parameters, and then the actual material parameters for the selected material model.
- The PolyUMod User's Manual contains the details.
- Read Chapter 3 in the User's Manual!

Copyright Veryst Engineering , 2015

PolyUMod Hints

- For most problems pick one of the material models highlighted in blue on Slide 9.
- Always activate non-linear geometry in the FE program.
- Allow the FE program to use additional attempts per iteration before cutting time.
- Some problems benefit from using a numerical Jacobian (JAC=3) instead of the default closed-form approximation.
- When using plane-stress or membrane elements you may need to set the global parameter TWOD_S.
- For anisotropic materials you may need to set the global parameter ORIENT.

Copyright Veryst Engineering , 2015

MCalibration Workflow

- The following tutorial shows how a set of experimental data can be read into MCalibration and then used to calibrate a material model.
- It is shown how the behavior of the calibrated material model can be examined using virtual experiments.
- It is also shown how the material model can be exported to a finite element program.

Copyright Veryst Engineering , 2015

Experimental Data

Tension_0_017_s.txt

```
0, 0, 0
9.78343195266, 0.0016534, 0.43281
19.7625248521, 0.0033398667, 0.76299
29.736883432, 0.0050255333, 1.04102
39.7203153846, 0.0067127333, 1.32088
49.7017751479, 0.0083996, 1.56923
59.6856017751, 0.0100868667, 1.79433
69.659
79.641
89.625
99.604
109.58
119.56
129.54
139.52
149.50
159.48
```

Tension_0_00017_s.txt

```
0, 0, 0
0.0600710059172, 0.0010152, 0.24612
0.17285995858, 0.0029213333, 0.82719
0.285644970414, 0.0048274, 1.37232
0.392958579882, 0.006641, 1.83245
0.496311639053, 0.0083876667, 2.23962
0.597660751479, 0.0101004667, 2.57608
0.69813412426, 0.0117984667, 2.92867
0.798161733728, 0.0134889333, 3.22346
0.898161733728, 0.0151789333, 3.48739
0.998023668639, 0.0168666, 3.75748
1.09786193491, 0.0185538667, 3.96155
1.19760552071, 0.0202395333, 4.18045
1.2973964497, 0.021926, 4.37969
1.39723471598, 0.0236132667, 4.56692
1.49700197041, 0.0252993333, 4.79743
1.59693491124, 0.0269882, 4.97169
```

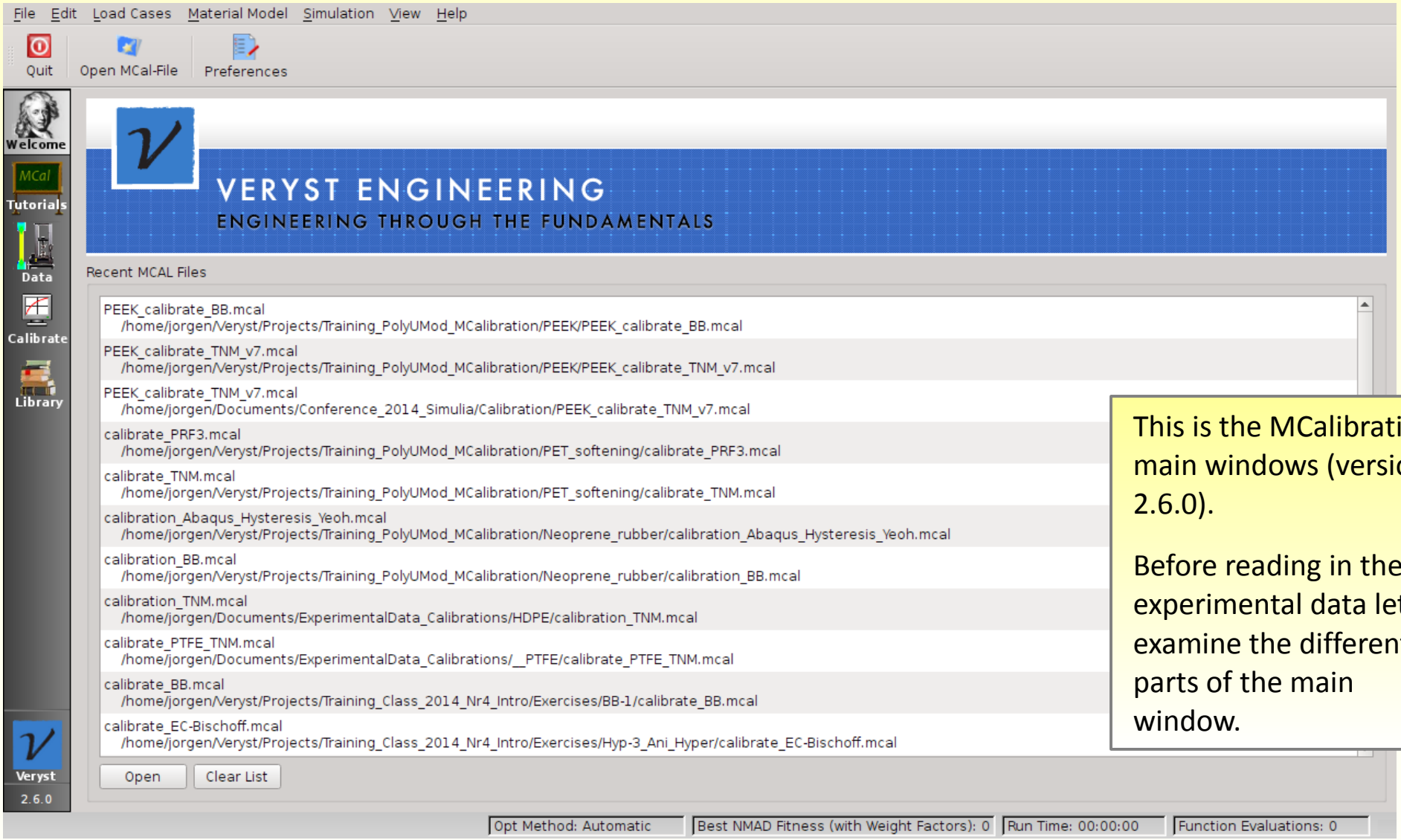
In this example we will use uniaxial tension data at two different strain rates (for a real application it may be useful to have additional experimental data).

The figures to the left show parts of the data files using a text editor.

The first step is to read these files into MCalibration.

Copyright Veryst Engineering , 2015

Experimental Data

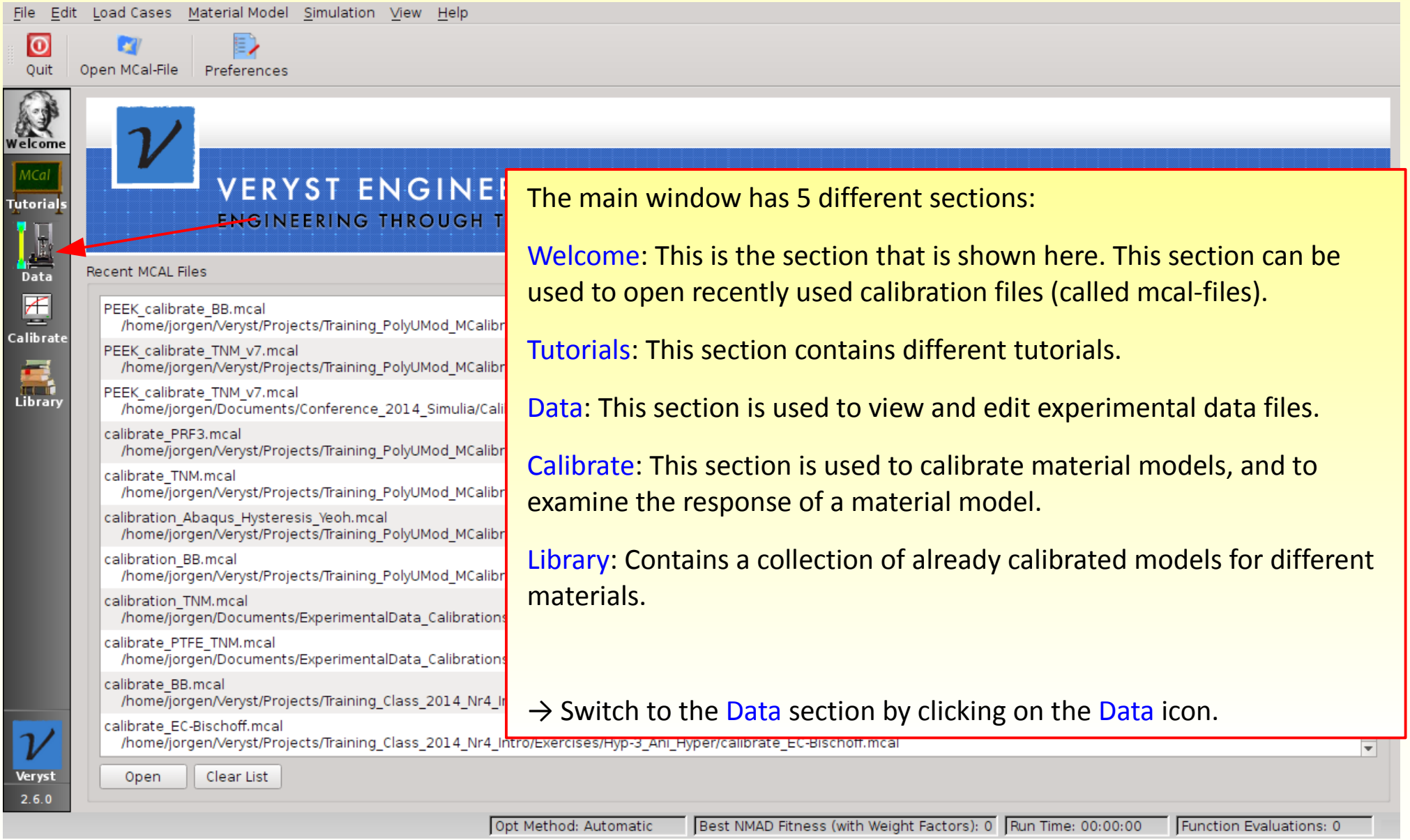


This is the MCalibration main windows (version 2.6.0).

Before reading in the experimental data let's examine the different parts of the main window.

Copyright Veryst Engineering , 2015

Experimental Data



The main window has 5 different sections:

Welcome: This is the section that is shown here. This section can be used to open recently used calibration files (called mcal-files).

Tutorials: This section contains different tutorials.

Data: This section is used to view and edit experimental data files.

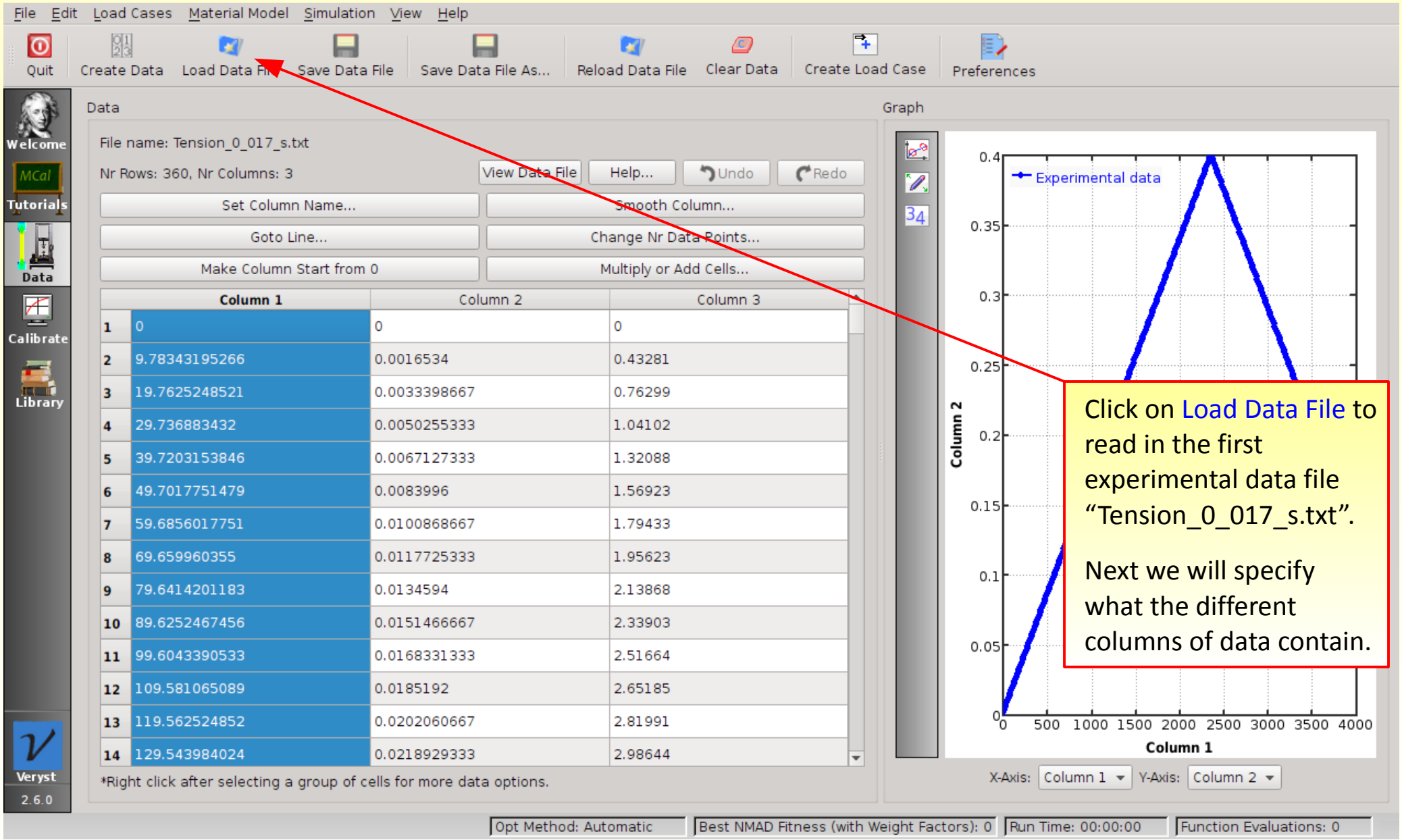
Calibrate: This section is used to calibrate material models, and to examine the response of a material model.

Library: Contains a collection of already calibrated models for different materials.

→ Switch to the Data section by clicking on the Data icon.

Copyright Veryst Engineering , 2015

Experimental Data

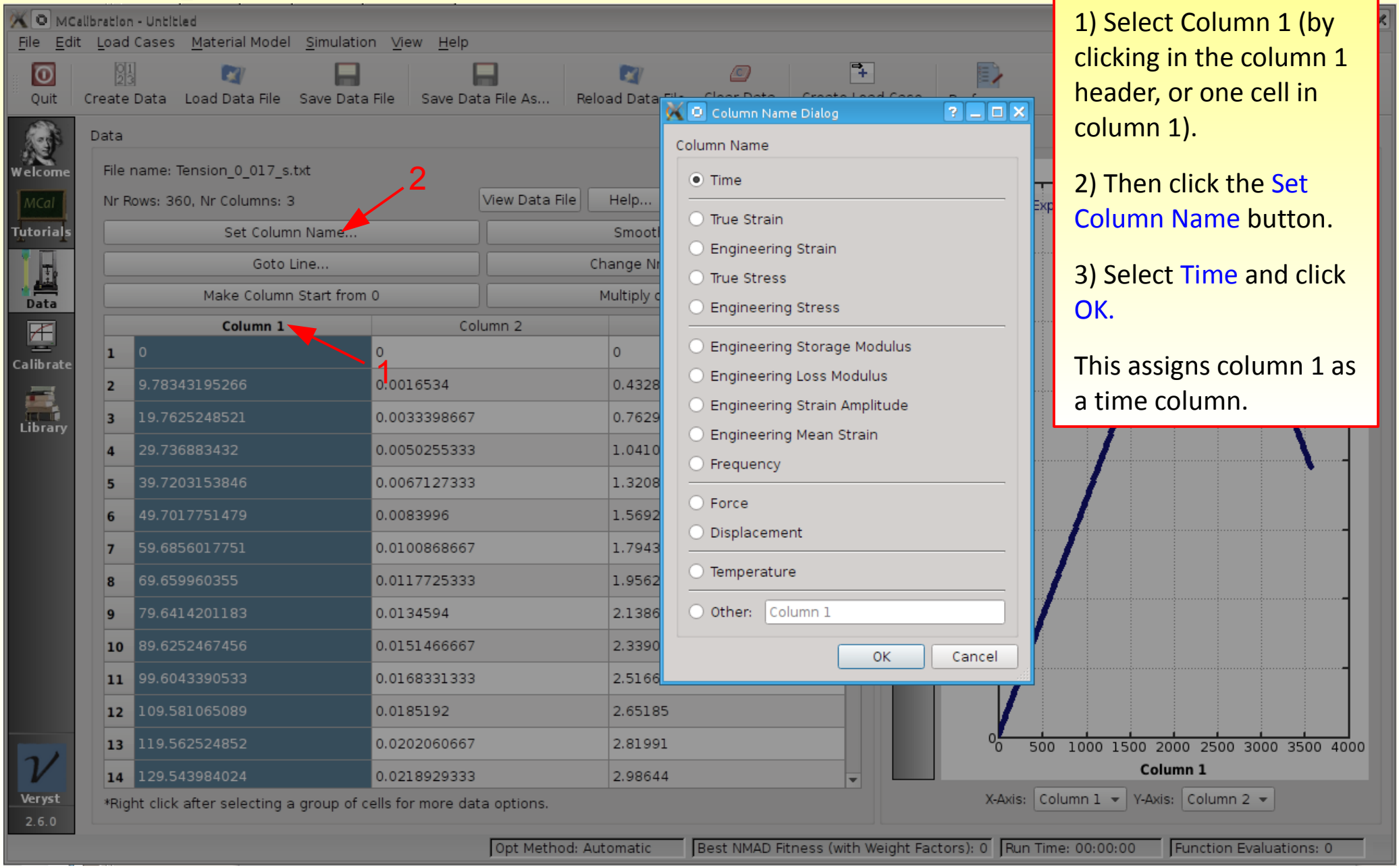


Click on Load Data File to read in the first experimental data file "Tension_0_017_s.txt".

Next we will specify what the different columns of data contain.

Copyright Veryst Engineering , 2015

Experimental Data



1) Select Column 1 (by clicking in the column 1 header, or one cell in column 1).

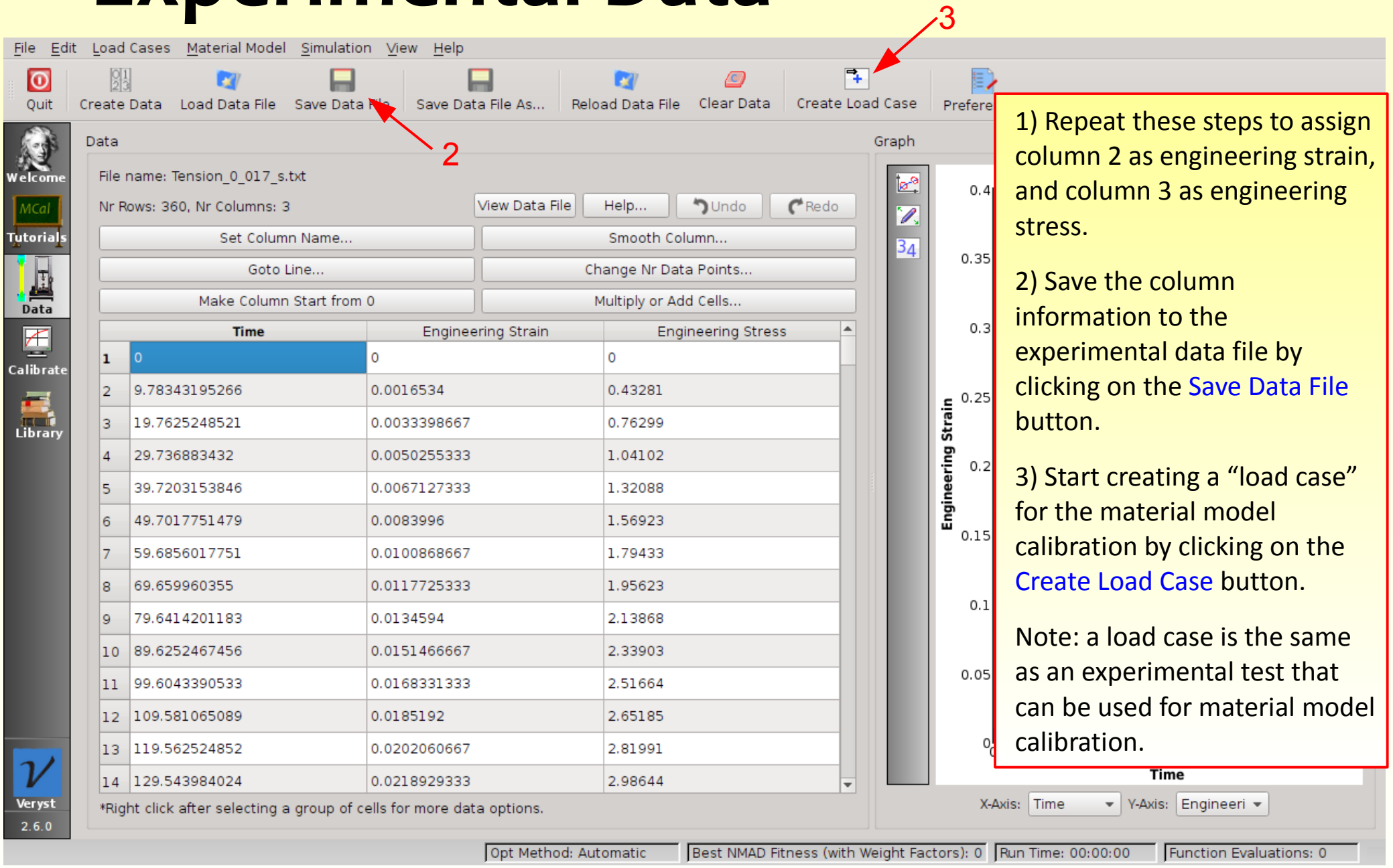
2) Then click the Set Column Name button.

3) Select Time and click OK.

This assigns column 1 as a time column.

Copyright Veryst Engineering , 2015

Experimental Data



1) Repeat these steps to assign column 2 as engineering strain, and column 3 as engineering stress.

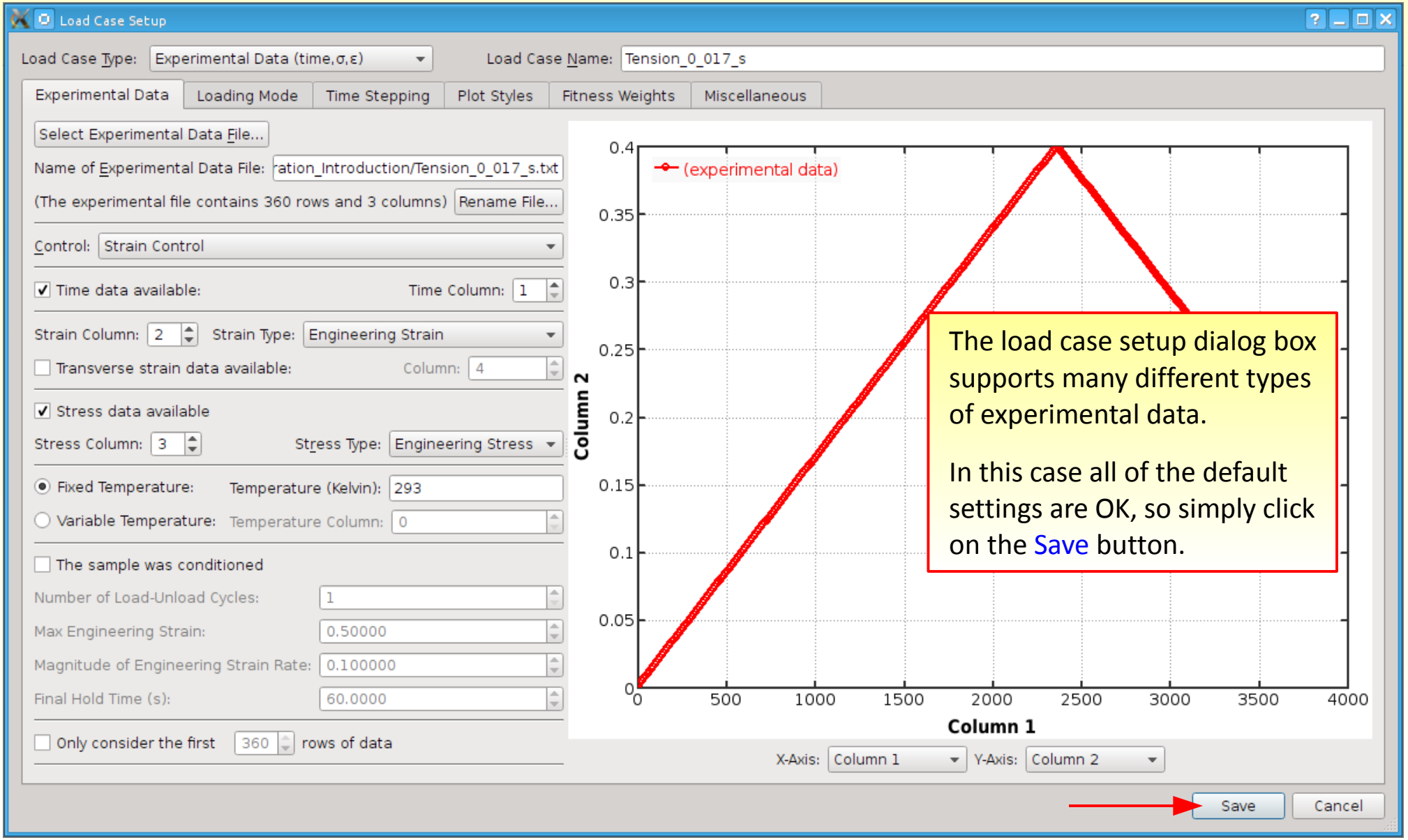
2) Save the column information to the experimental data file by clicking on the Save Data File button.

3) Start creating a “load case” for the material model calibration by clicking on the Create Load Case button.

Note: a load case is the same as an experimental test that can be used for material model calibration.

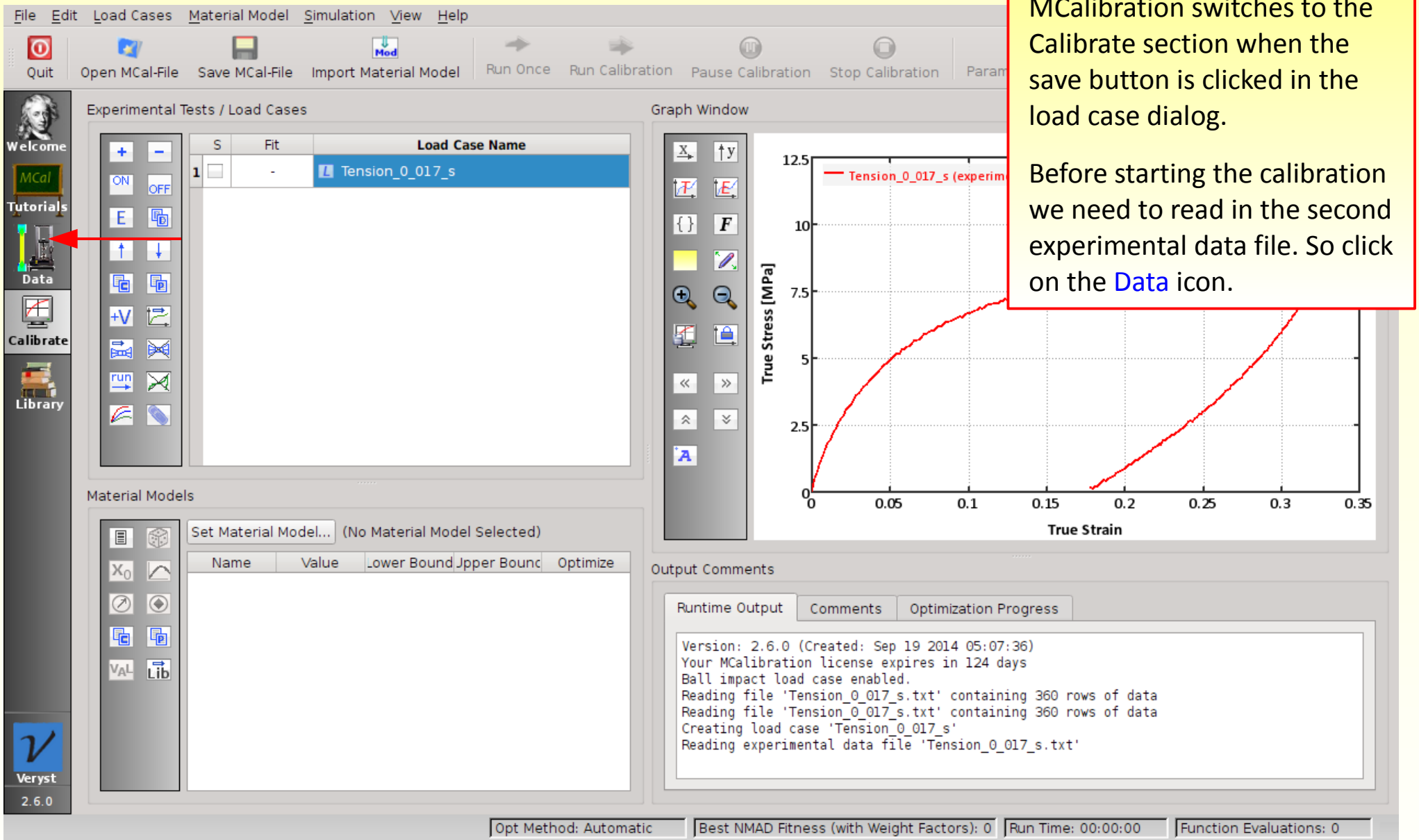
Copyright Veryst Engineering , 2015

Experimental Data



Copyright Veryst Engineering , 2015

Experimental Data

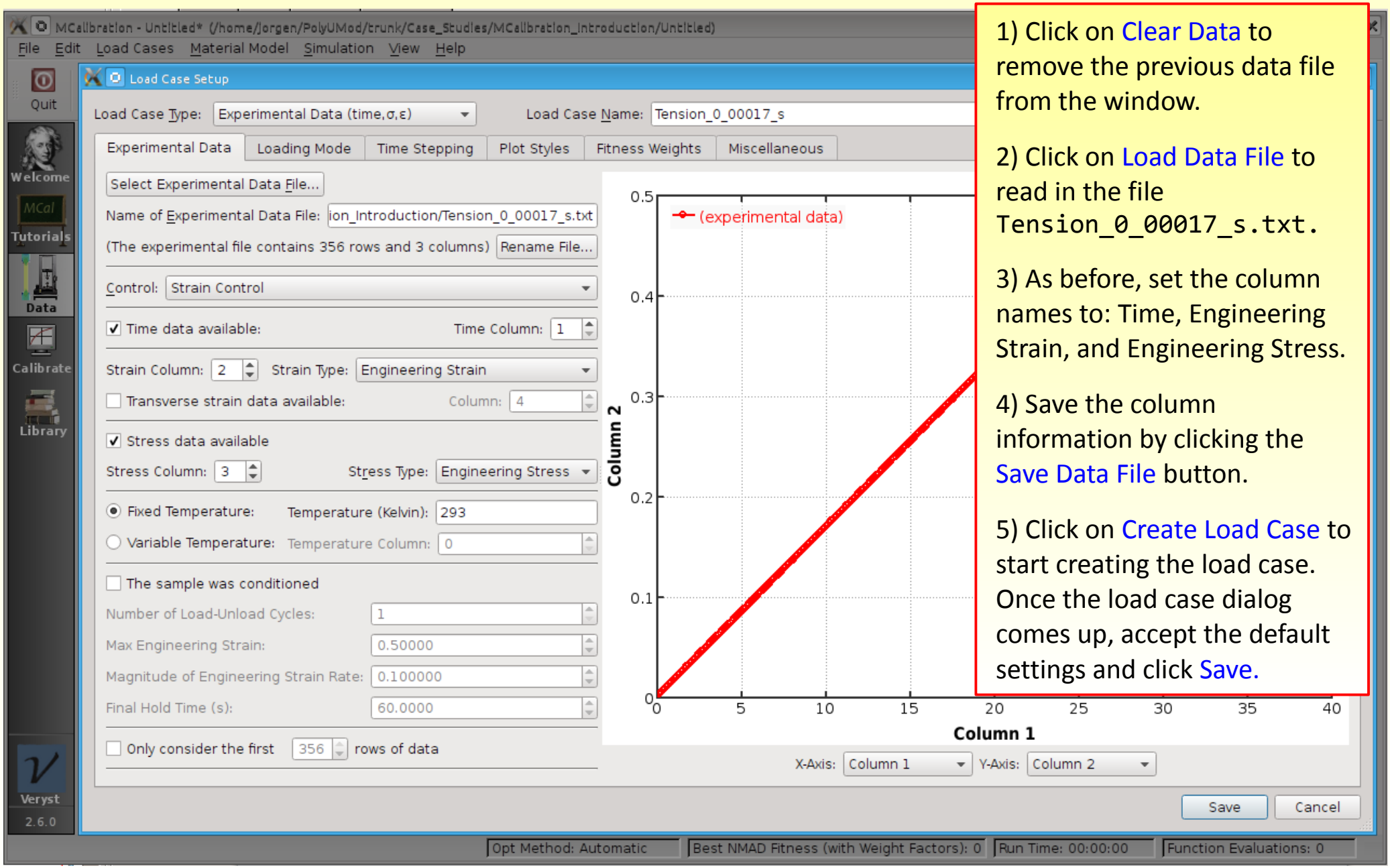


MCalibration switches to the Calibrate section when the save button is clicked in the load case dialog.

Before starting the calibration we need to read in the second experimental data file. So click on the Data icon.

Copyright Veryst Engineering , 2015

Experimental Data



1) Click on Clear Data to remove the previous data file from the window.

2) Click on Load Data File to read in the file `Tension_0_00017_s.txt`.

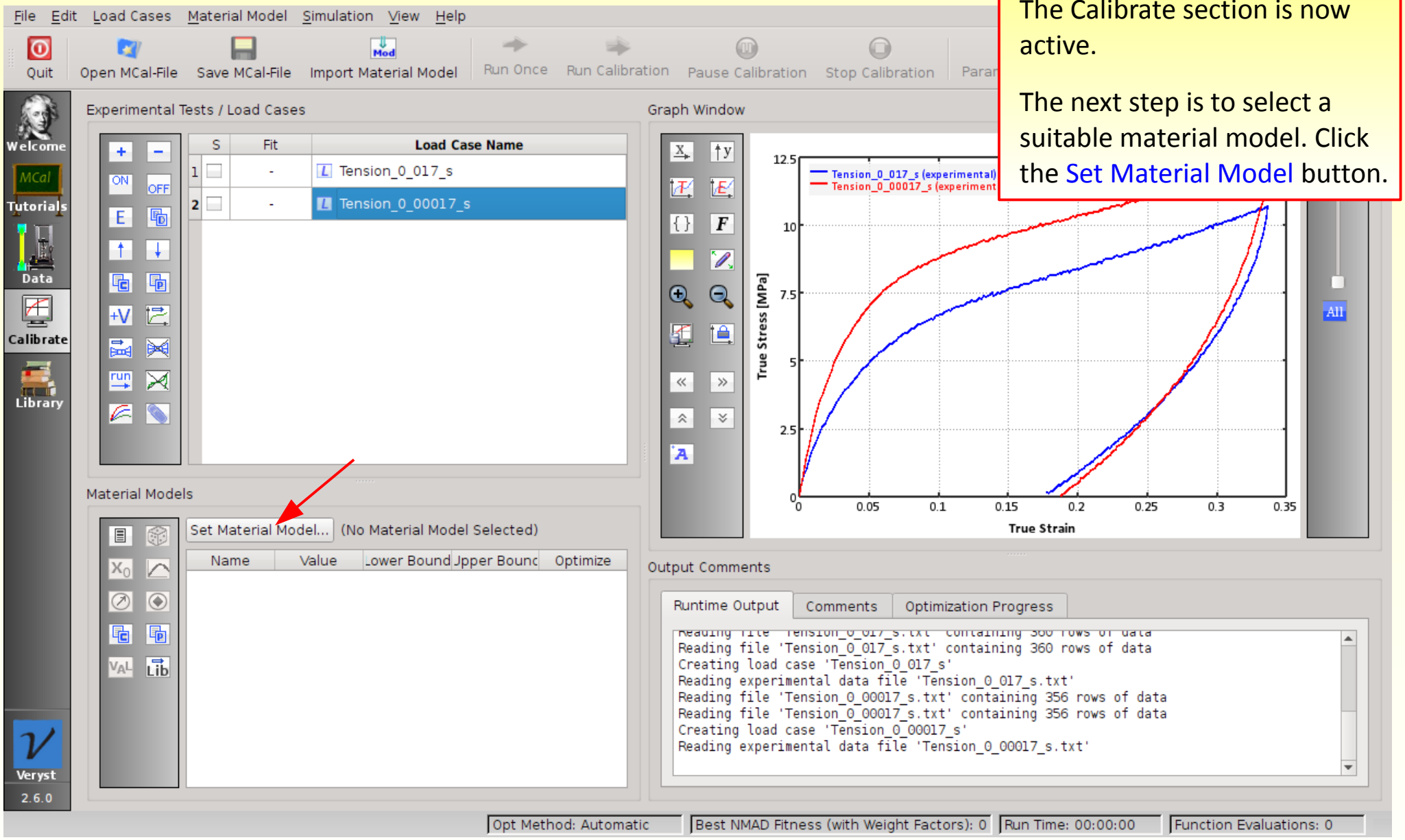
3) As before, set the column names to: Time, Engineering Strain, and Engineering Stress.

4) Save the column information by clicking the Save Data File button.

5) Click on Create Load Case to start creating the load case. Once the load case dialog comes up, accept the default settings and click Save.

Copyright Veryst Engineering , 2015

Experimental Data

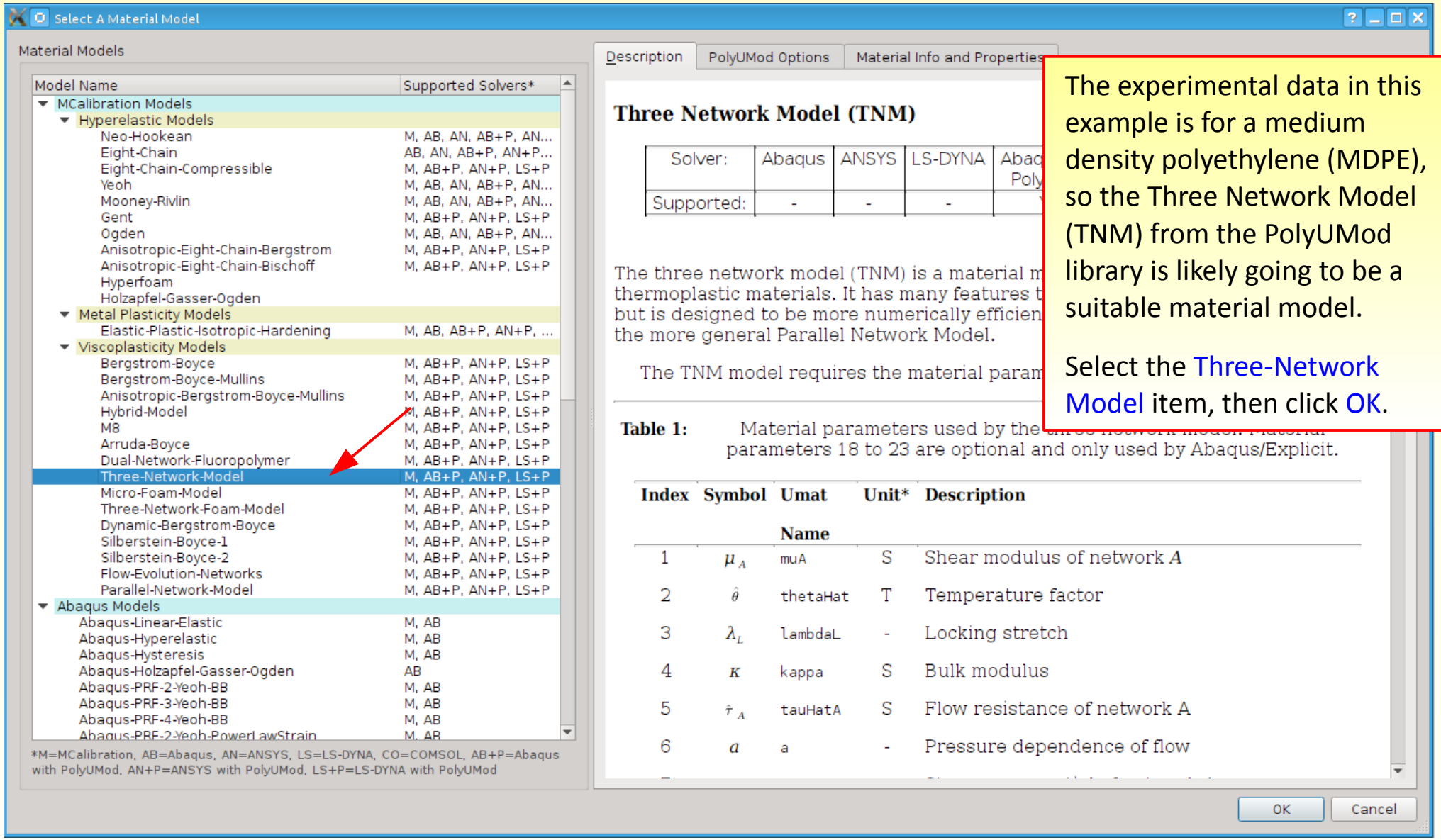


The Calibrate section is now active.

The next step is to select a suitable material model. Click the Set Material Model button.

Copyright Veryst Engineering , 2015

Material Model

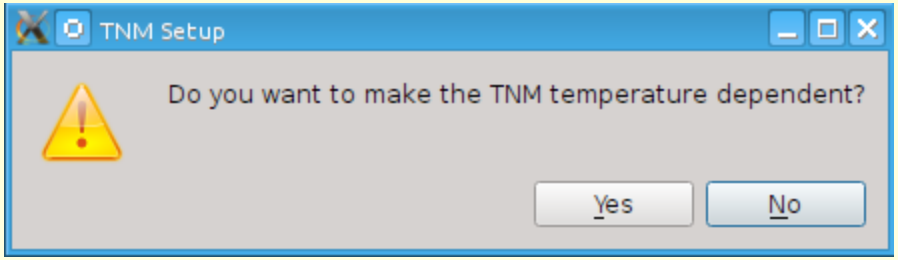

Select A Material Model
Material Models
Model Name
Supported Solvers*
MCalibration Models
Hyperelastic Models
Neo-Hookean M, AB, AN, AB+P, AN...
Eight-Chain AB, AN, AB+P, AN+P...
Eight-Chain-Compressible M, AB+P, AN+P, LS+P
Yeoh M, AB, AN, AB+P, AN...
Mooney-Rivlin M, AB, AN, AB+P, AN...
Gent M, AB+P, AN+P, LS+P
Ogden M, AB, AN, AB+P, AN...
Anisotropic-Eight-Chain-Bergstrom M, AB+P, AN+P, LS+P
Anisotropic-Eight-Chain-Bischoff M, AB+P, AN+P, LS+P
Hyperfoam
Holzapfel-Gasser-Ogden
Metal Plasticity Models
Elastic-Plastic-Isotropic-Hardening M, AB, AB+P, AN+P, ...
Viscoplasticity Models
Bergstrom-Boyce M, AB+P, AN+P, LS+P
Bergstrom-Boyce-Mullins M, AB+P, AN+P, LS+P
Anisotropic-Bergstrom-Boyce-Mullins M, AB+P, AN+P, LS+P
Hybrid-Model M, AB+P, AN+P, LS+P
M8 M, AB+P, AN+P, LS+P
Arruda-Boyce M, AB+P, AN+P, LS+P
Dual-Network-Fluoropolymer M, AB+P, AN+P, LS+P
Three-Network-Model M, AB+P, AN+P, LS+P
Micro-Foam-Model M, AB+P, AN+P, LS+P
Three-Network-Foam-Model M, AB+P, AN+P, LS+P
Dynamic-Bergstrom-Boyce M, AB+P, AN+P, LS+P
Silberstein-Boyce-1 M, AB+P, AN+P, LS+P
Silberstein-Boyce-2 M, AB+P, AN+P, LS+P
Flow-Evolution-Networks M, AB+P, AN+P, LS+P
Parallel-Network-Model M, AB+P, AN+P, LS+P
Abaqus Models
Abaqus-Linear-Elastic M, AB
Abaqus-Hyperelastic M, AB
Abaqus-Hysteresis M, AB
Abaqus-Holzapfel-Gasser-Ogden AB
Abaqus-PRF-2-Yeoh-BB M, AB
Abaqus-PRF-3-Yeoh-BB M, AB
Abaqus-PRF-4-Yeoh-BB M, AB
*M=MCalibration, AB=Abaqus, AN=ANSYS, LS=LS-DYNA, CO=COMSOL, AB+P=Abaqus with PolyUMod, AN+P=ANSYS with PolyUMod, LS+P=LS-DYNA with PolyUMod
Description
PolyUMod Options
Material Info and Properties
Three Network Model (TNM)
Solver: Abaqus ANSYS LS-DYNA
Supported: - - -
The three network model (TNM) is a material m
thermoplastic materials. It has many features t
but is designed to be more numerically efficien
the more general Parallel Network Model.
The TNM model requires the material param
Table 1: Material parameters used by the
parameters 18 to 23 are optional and only used by Abaqus/Explicit.
Index Symbol Umat Name Unit* Description
1 μ_A muA S Shear modulus of network A
2 $\hat{\theta}$ thetaHat T Temperature factor
3 λ_L lambdaL - Locking stretch
4 κ kappa S Bulk modulus
5 $\hat{\tau}_A$ tauHatA S Flow resistance of network A
6 a a - Pressure dependence of flow
OK
Cancel

The experimental data in this example is for a medium density polyethylene (MDPE), so the Three Network Model (TNM) from the PolyUMod library is likely going to be a suitable material model.

Select the Three-Network Model item, then click OK.

Copyright Veryst Engineering , 2015

Material Model

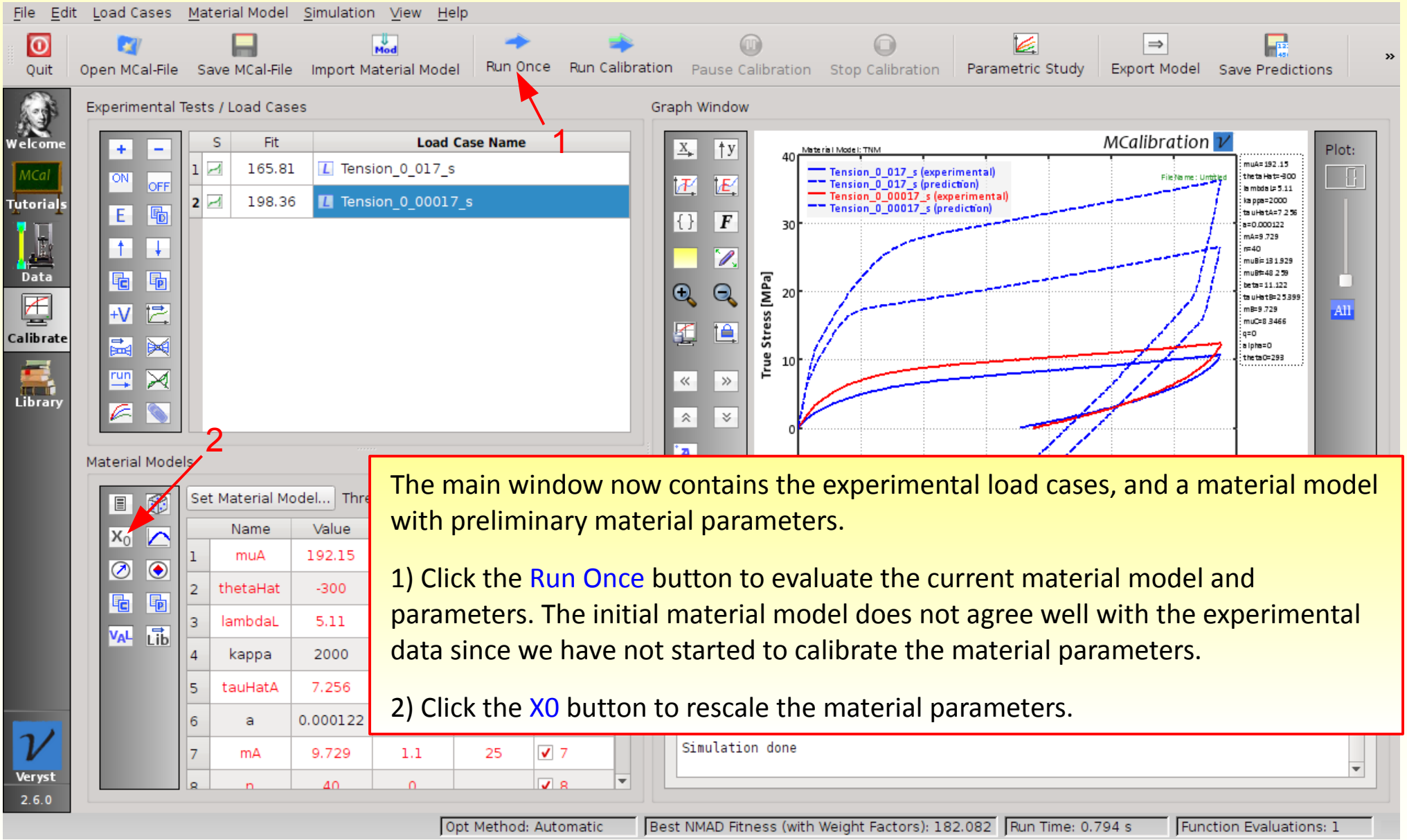


MCalibration then asks if the Three Network model should be made temperature dependent.

In this case we select NO since the experimental data was only at room temperature.

Copyright Veryst Engineering , 2015

Preliminary Calibration



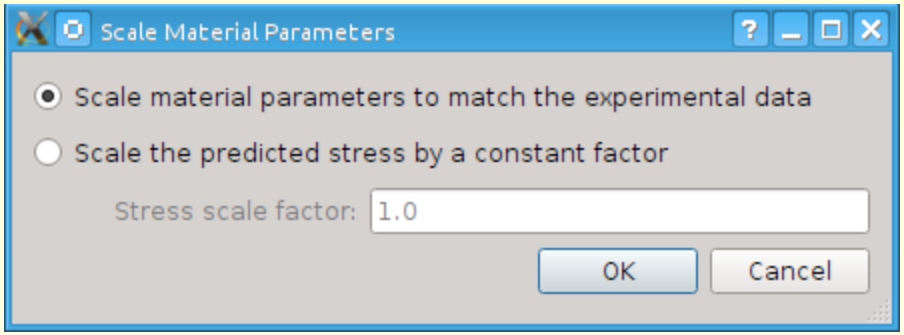
The main window now contains the experimental load cases, and a material model with preliminary material parameters.

1) Click the Run Once button to evaluate the current material model and parameters. The initial material model does not agree well with the experimental data since we have not started to calibrate the material parameters.

2) Click the X0 button to rescale the material parameters.

Copyright Veryst Engineering , 2015

Preliminary Calibration

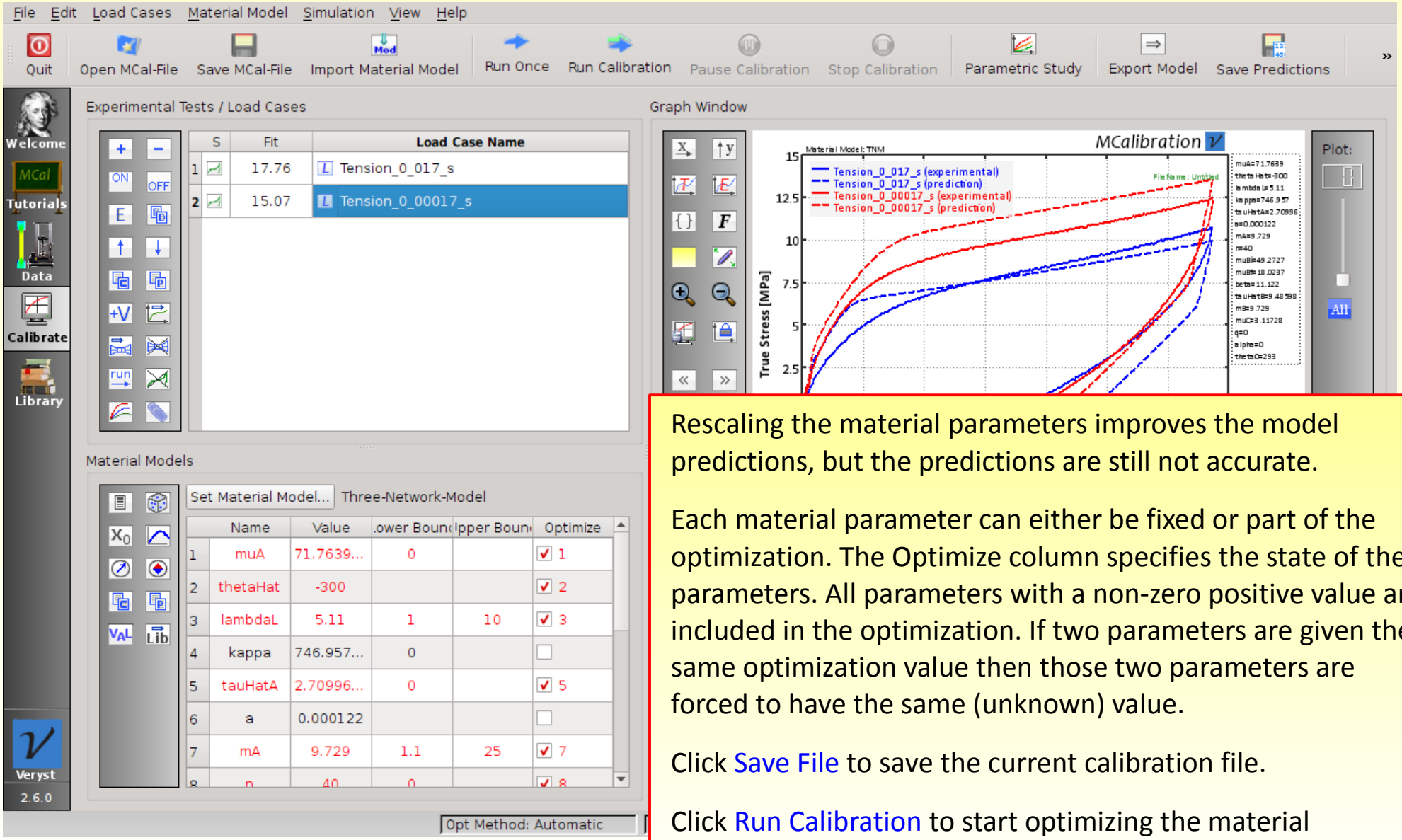


In this case the model predicted stresses were about 4 times too high.

By selecting Scale material parameters to match the experimental data, and clicking OK, MCalibration will automatically rescale all material parameters with units of stress so that the material model predictions approximately match the maximum experimental stress.

Copyright Veryst Engineering , 2015

Model Calibration



Rescaling the material parameters improves the model predictions, but the predictions are still not accurate.

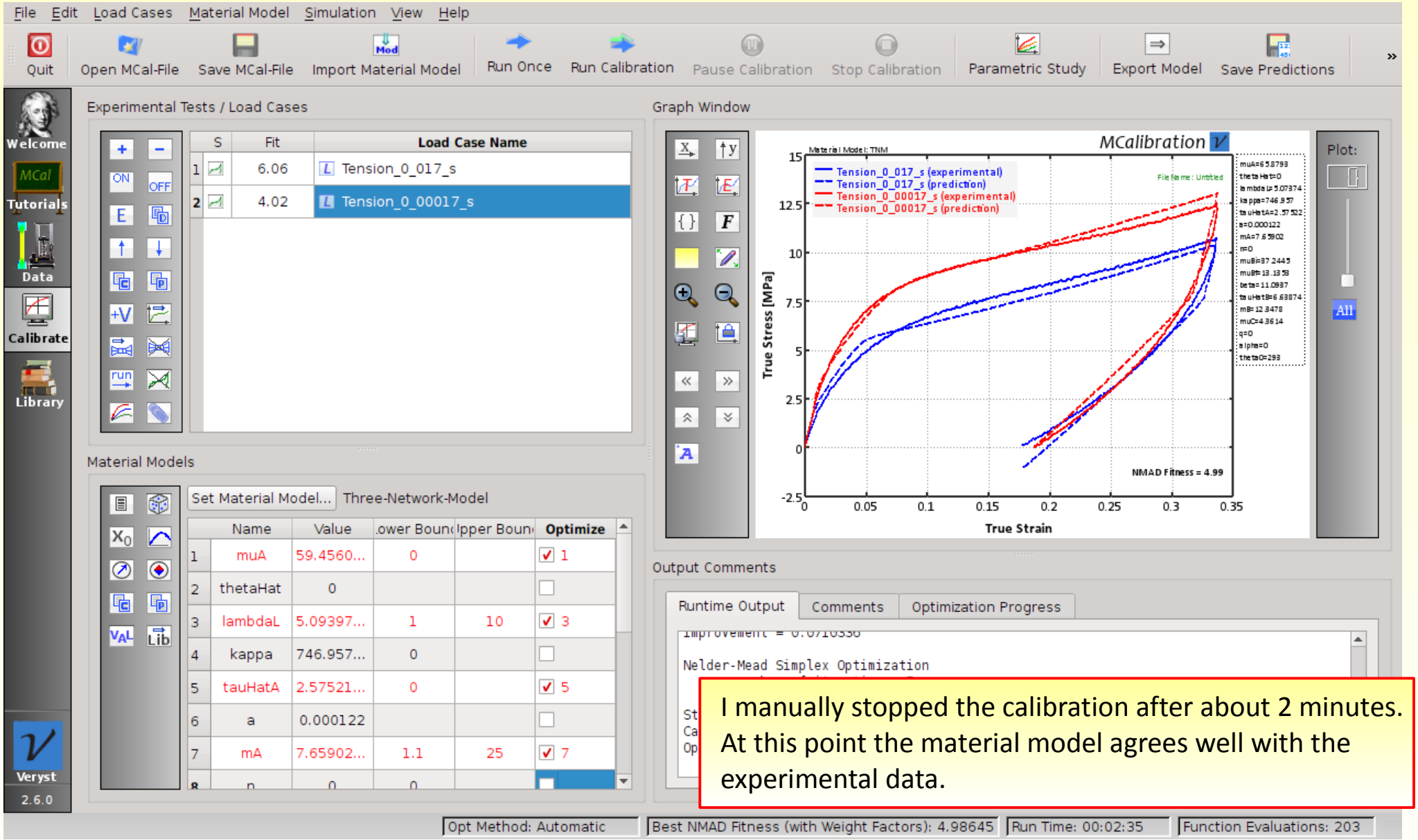
Each material parameter can either be fixed or part of the optimization. The Optimize column specifies the state of the parameters. All parameters with a non-zero positive value are included in the optimization. If two parameters are given the same optimization value then those two parameters are forced to have the same (unknown) value.

Click Save File to save the current calibration file.

Click Run Calibration to start optimizing the material parameters.

Copyright Veryst Engineering , 2015

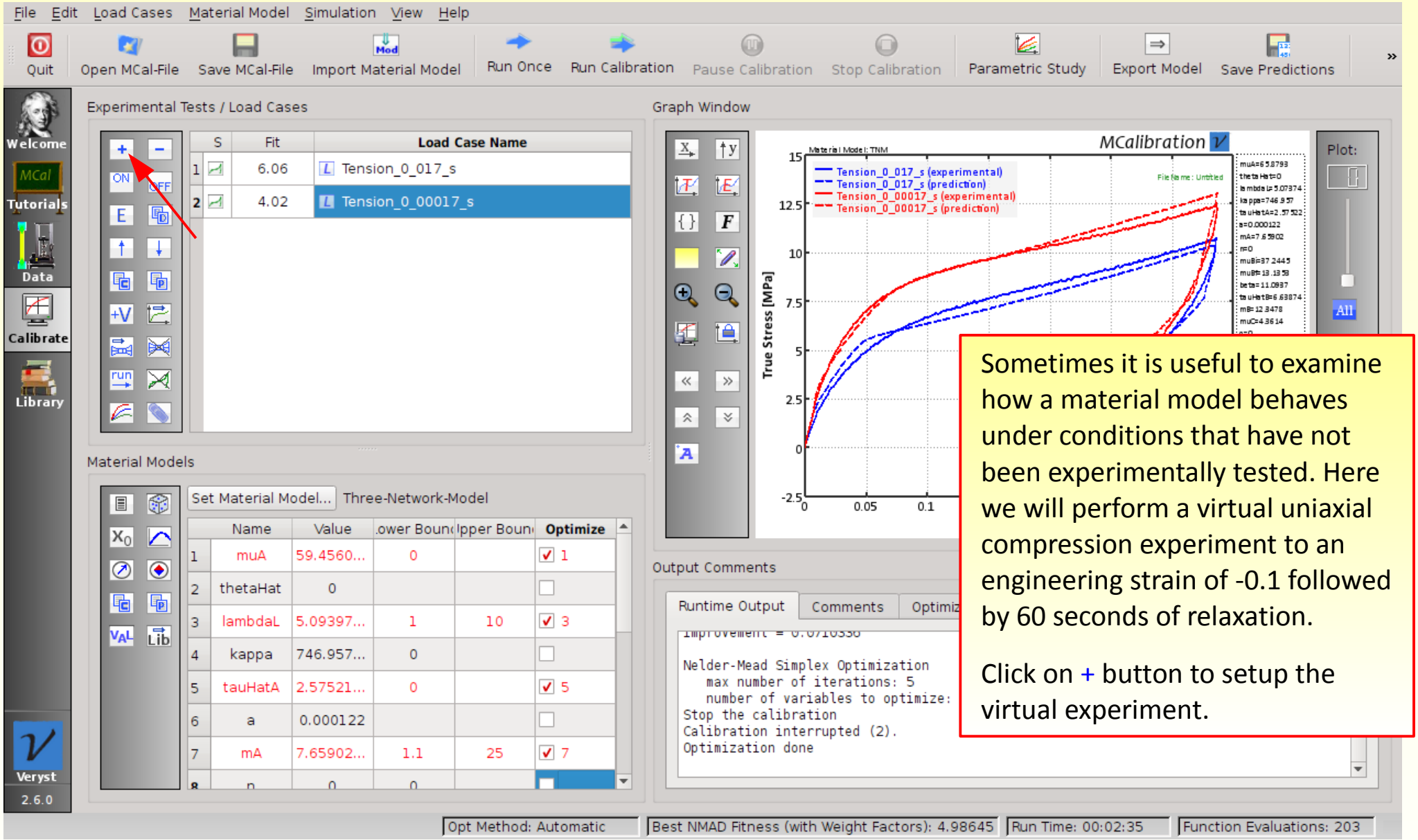
Model Calibration



I manually stopped the calibration after about 2 minutes. At this point the material model agrees well with the experimental data.

Copyright Veryst Engineering , 2015

Virtual Stress Relaxation

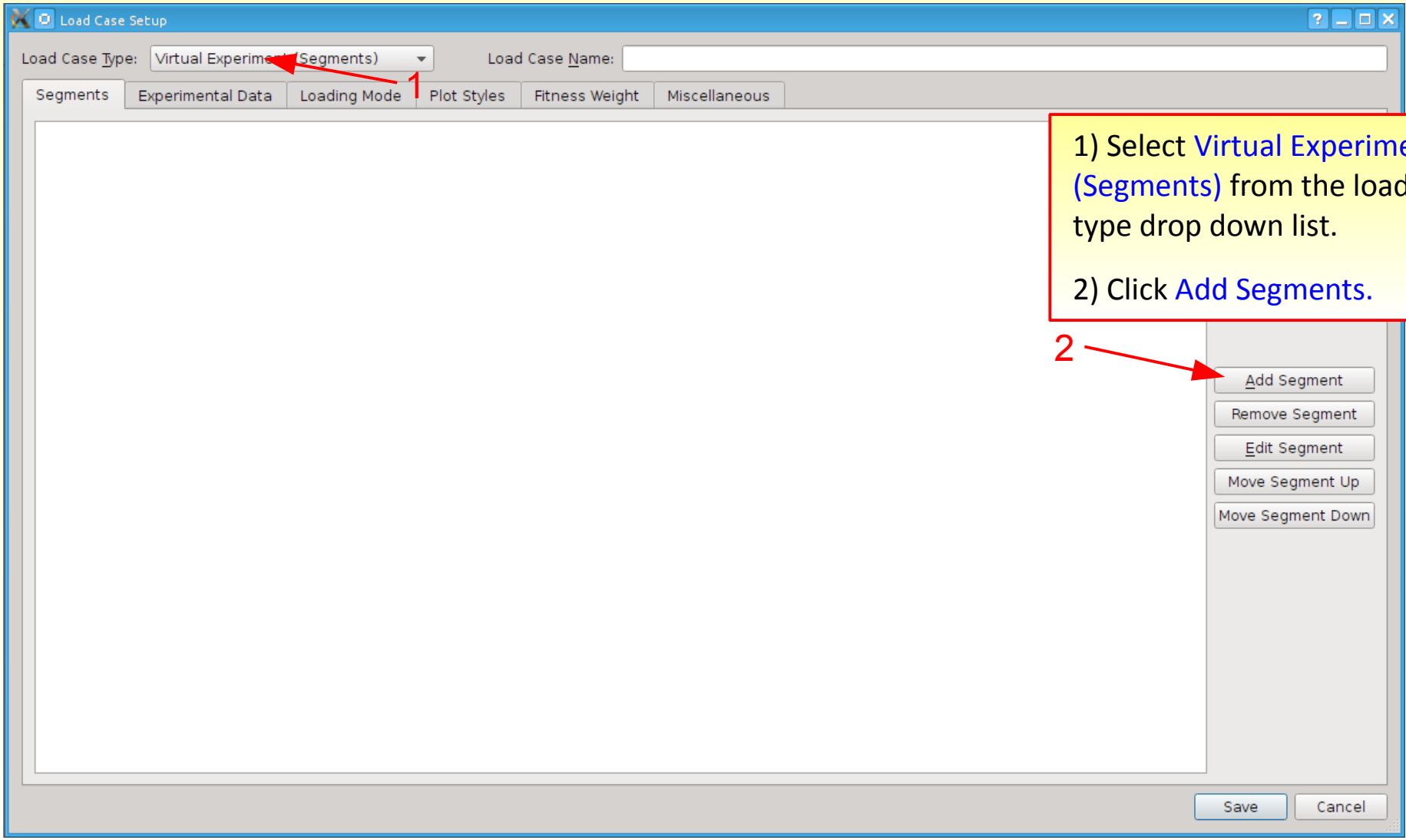


Sometimes it is useful to examine how a material model behaves under conditions that have not been experimentally tested. Here we will perform a virtual uniaxial compression experiment to an engineering strain of -0.1 followed by 60 seconds of relaxation.

Click on + button to setup the virtual experiment.

Copyright Veryst Engineering , 2015

Virtual Stress Relaxation

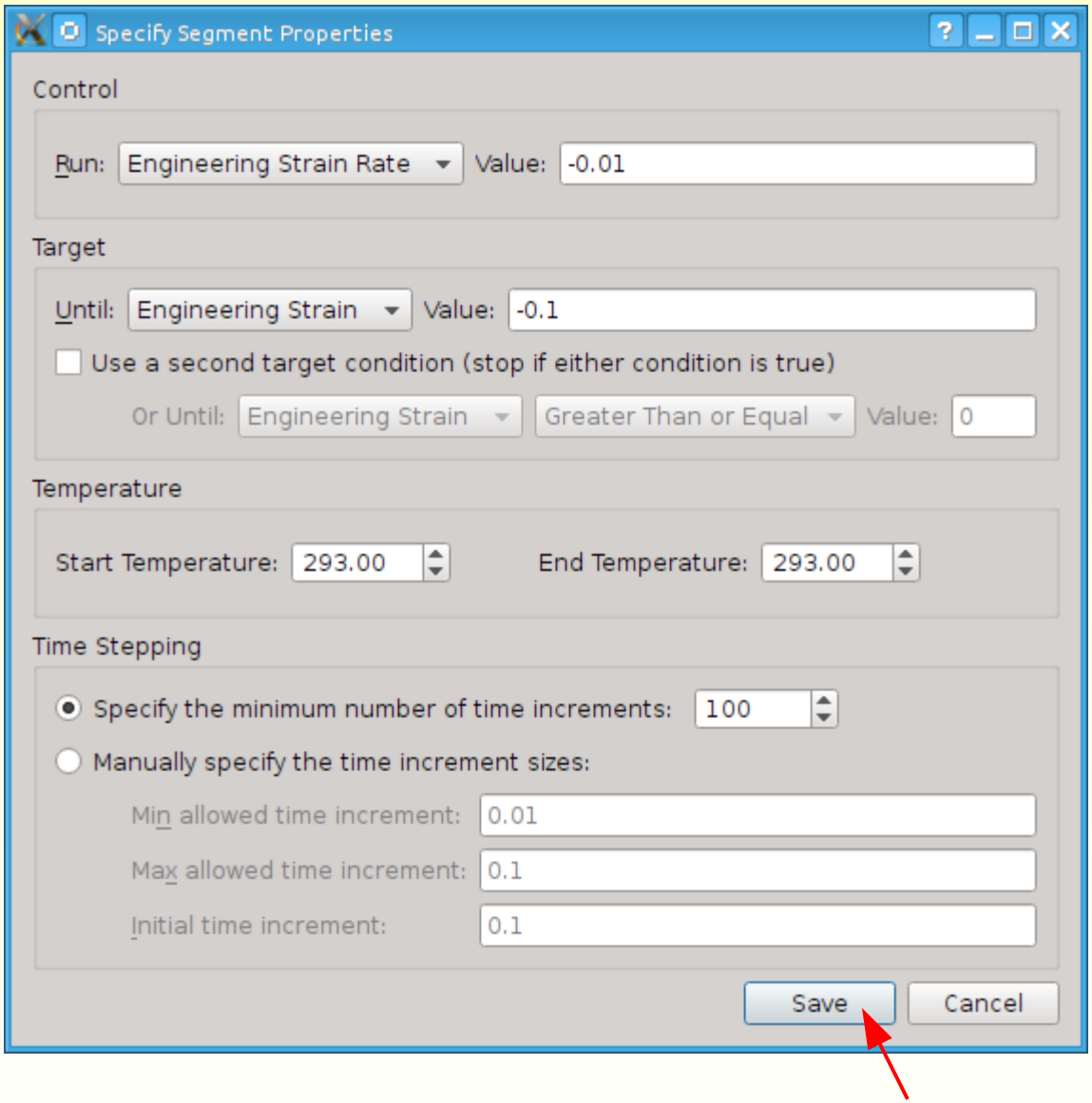


1) Select Virtual Experiment (Segments) from the load case type drop down list.

2) Click Add Segments.

Copyright Veryst Engineering , 2015

Virtual Stress Relaxation

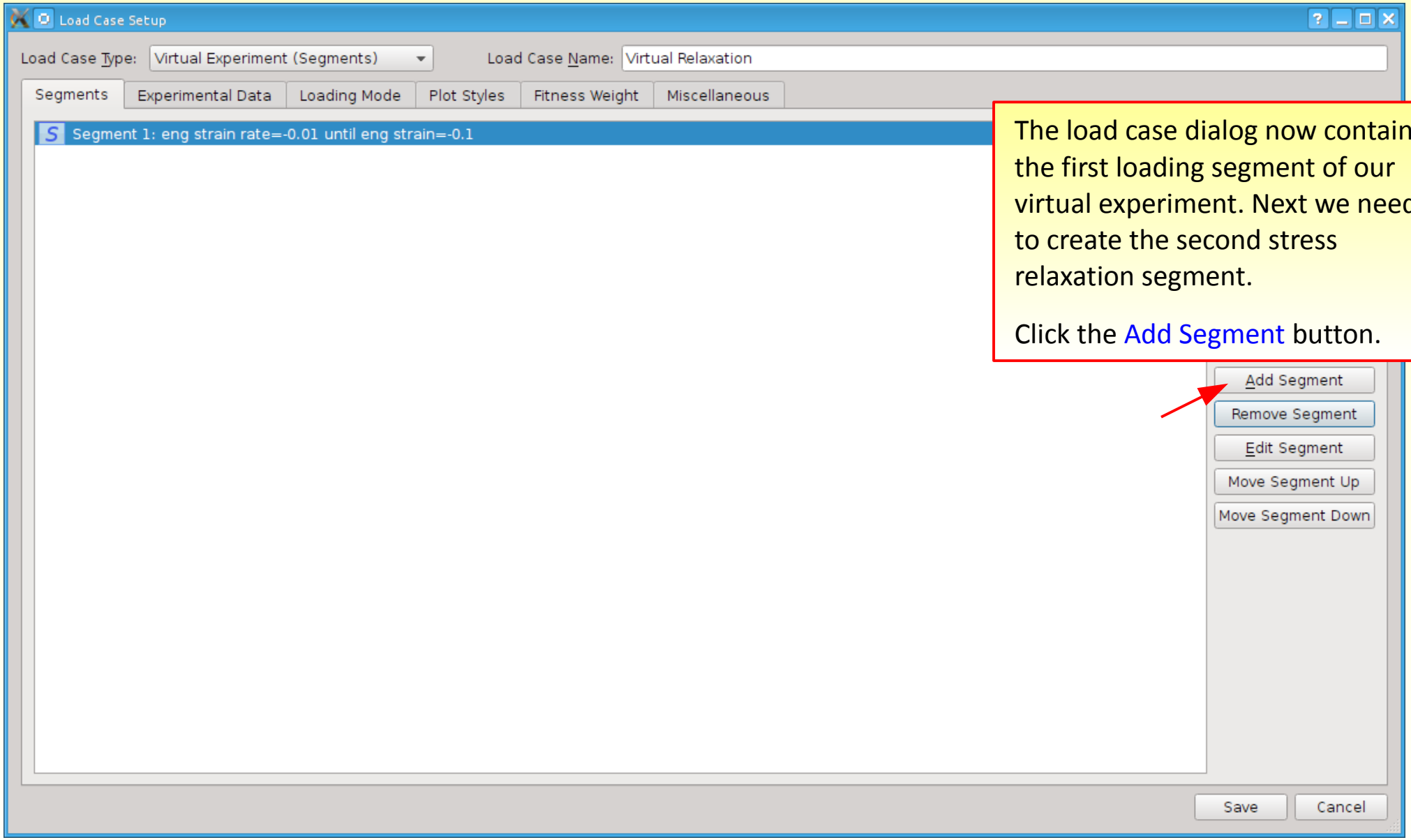

Specify Segment Properties
Control
Run: Engineering Strain Rate Value: -0.01
Target
Until: Engineering Strain Value: -0.1
Use a second target condition (stop if either condition is true)
Or Until: Engineering Strain Greater Than or Equal Value: 0
Temperature
Start Temperature: 293.00
End Temperature: 293.00
Time Stepping
Specify the minimum number of time increments: 100
Manually specify the time increment sizes:
Min allowed time increment: 0.01
Max allowed time increment: 0.1
Initial time increment: 0.1
Save
Cancel

Specify the target strain rate and strain value. This specifies the stress relaxation pre-load.

Then click the Save button.

Copyright Veryst Engineering , 2015

Virtual Stress Relaxation

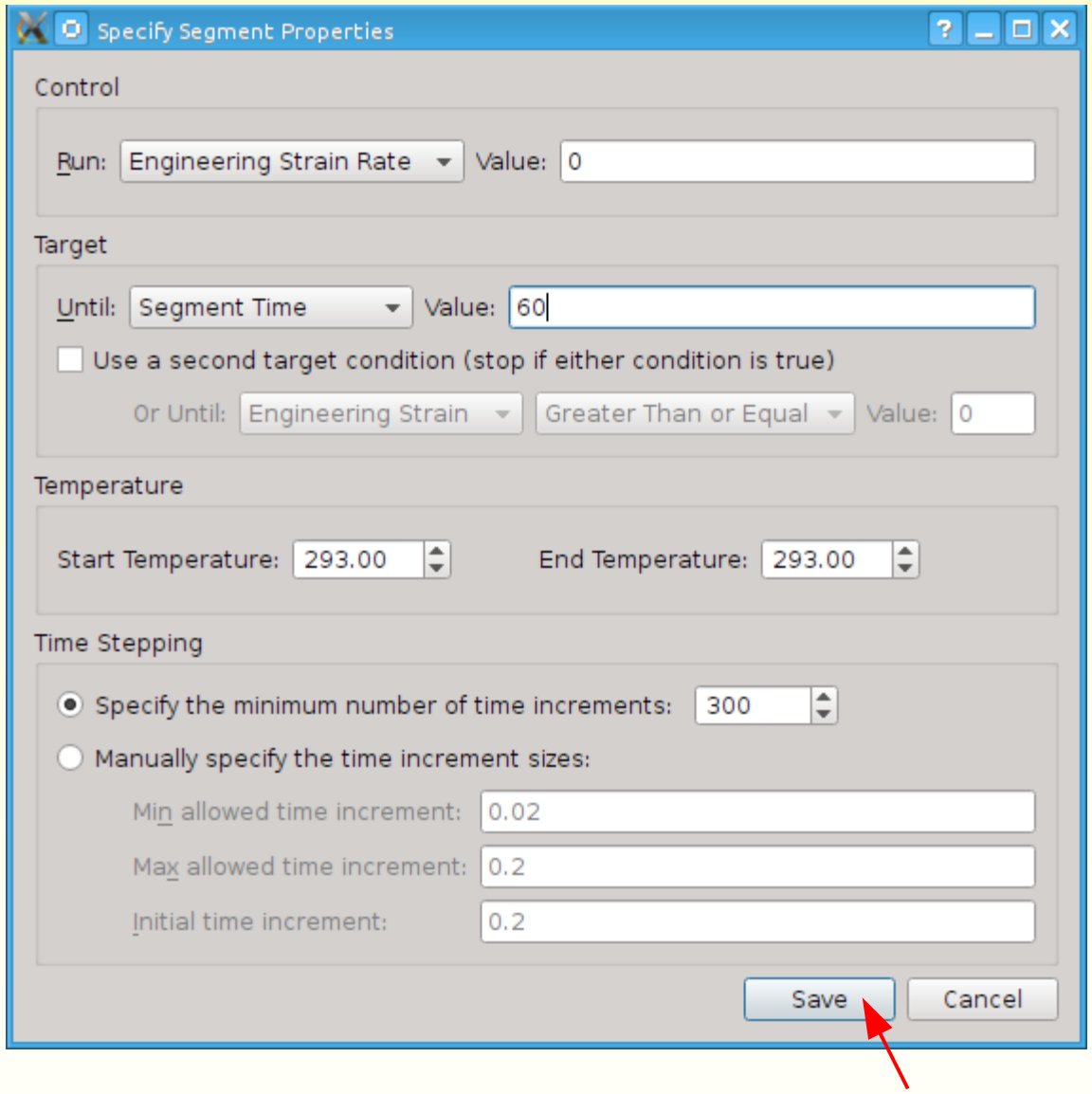


The load case dialog now contains the first loading segment of our virtual experiment. Next we need to create the second stress relaxation segment.

Click the Add Segment button.

Copyright Veryst Engineering , 2015

Virtual Stress Relaxation

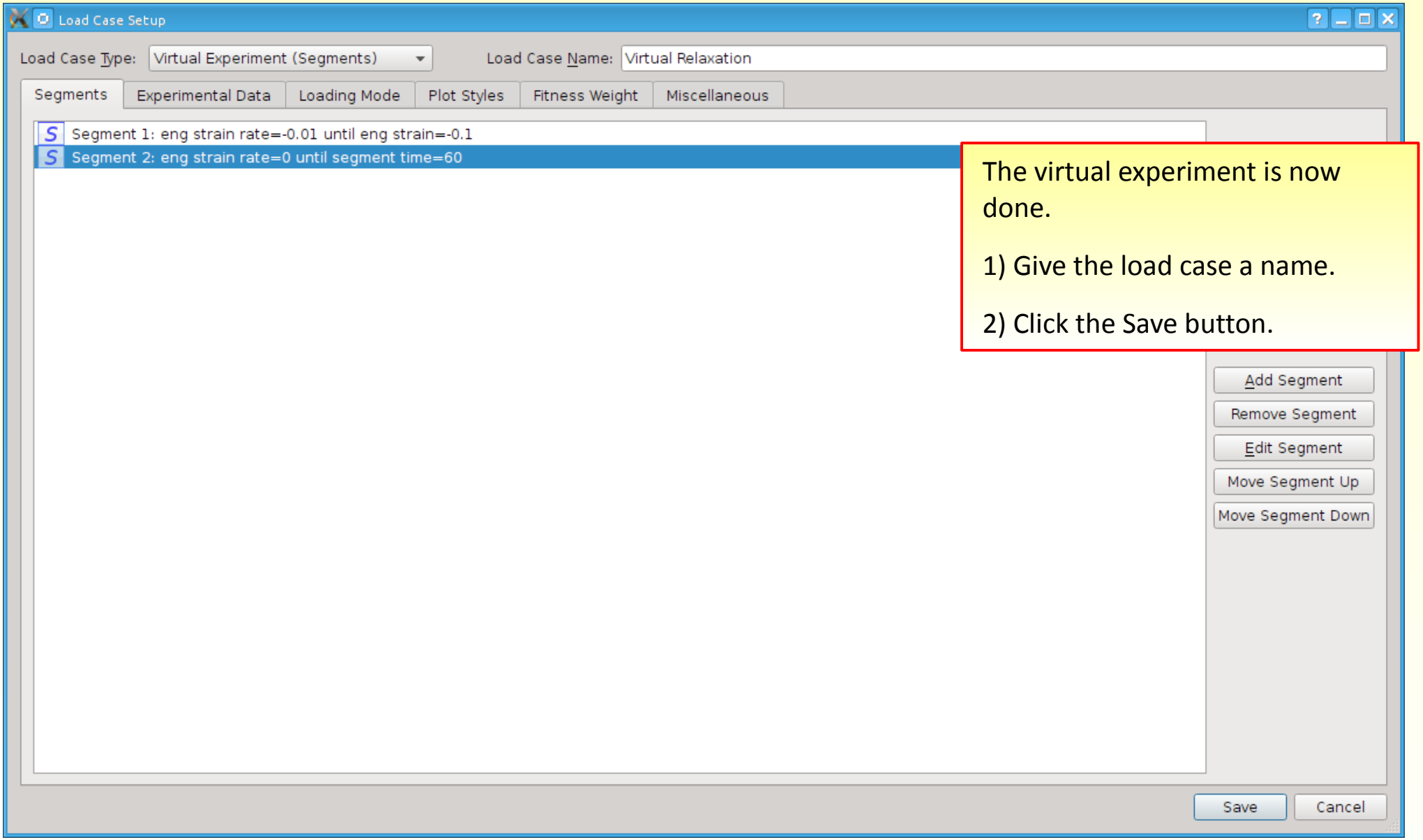

Specify Segment Properties
Control
Run: Engineering Strain Rate Value: 0
Target
Until: Segment Time Value: 60
Use a second target condition (stop if either condition is true)
Or Until: Engineering Strain Greater Than or Equal Value: 0
Temperature
Start Temperature: 293.00
End Temperature: 293.00
Time Stepping
Specify the minimum number of time increments: 300
Manually specify the time increment sizes:
Min allowed time increment: 0.02
Max allowed time increment: 0.2
Initial time increment: 0.2
Save
Cancel

The second loading segment has constant strain for 60 seconds.

Click Save.

Copyright Veryst Engineering , 2015

Virtual Stress Relaxation



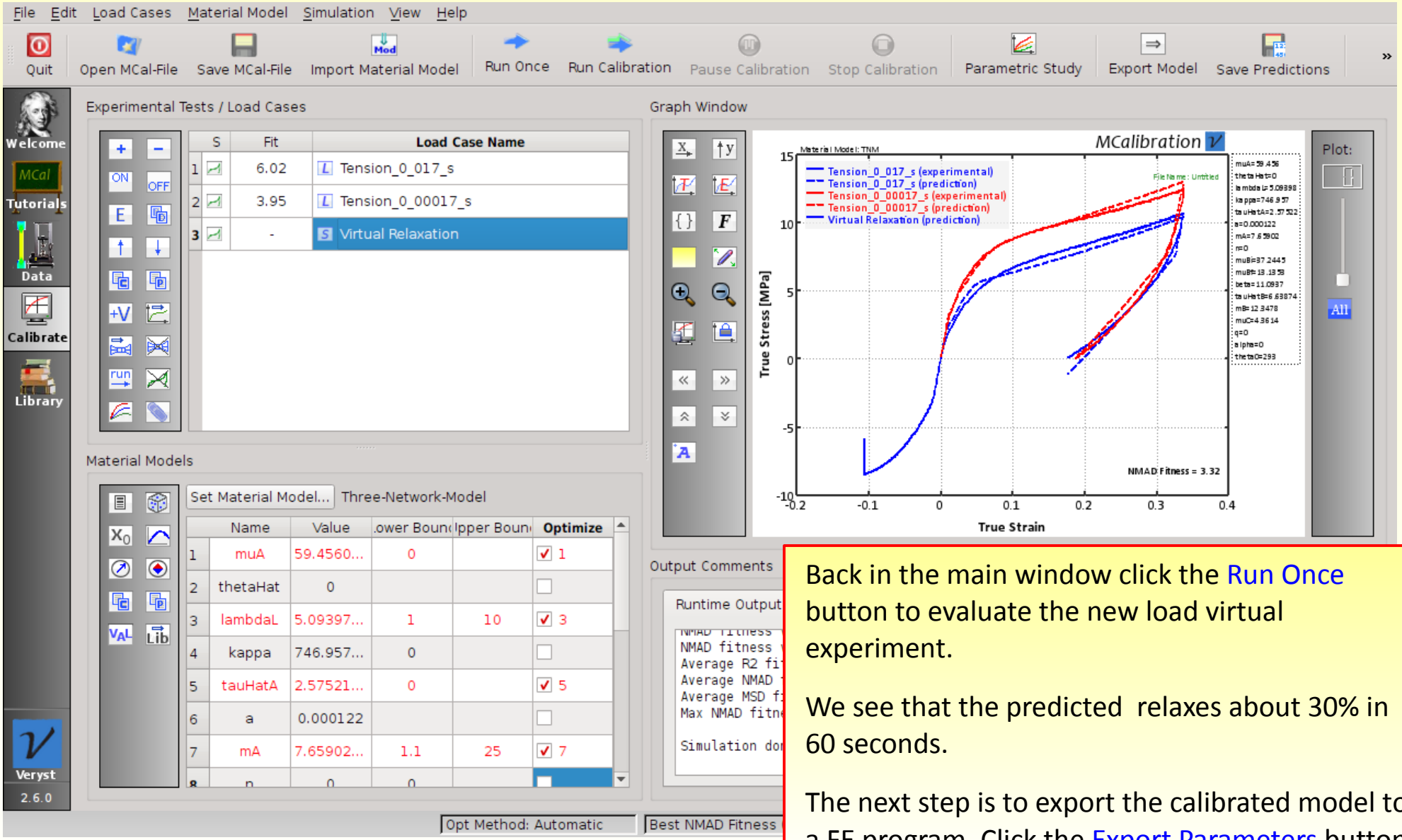
The virtual experiment is now done.

1) Give the load case a name.

2) Click the Save button.

Copyright Veryst Engineering , 2015

Virtual Stress Relaxation



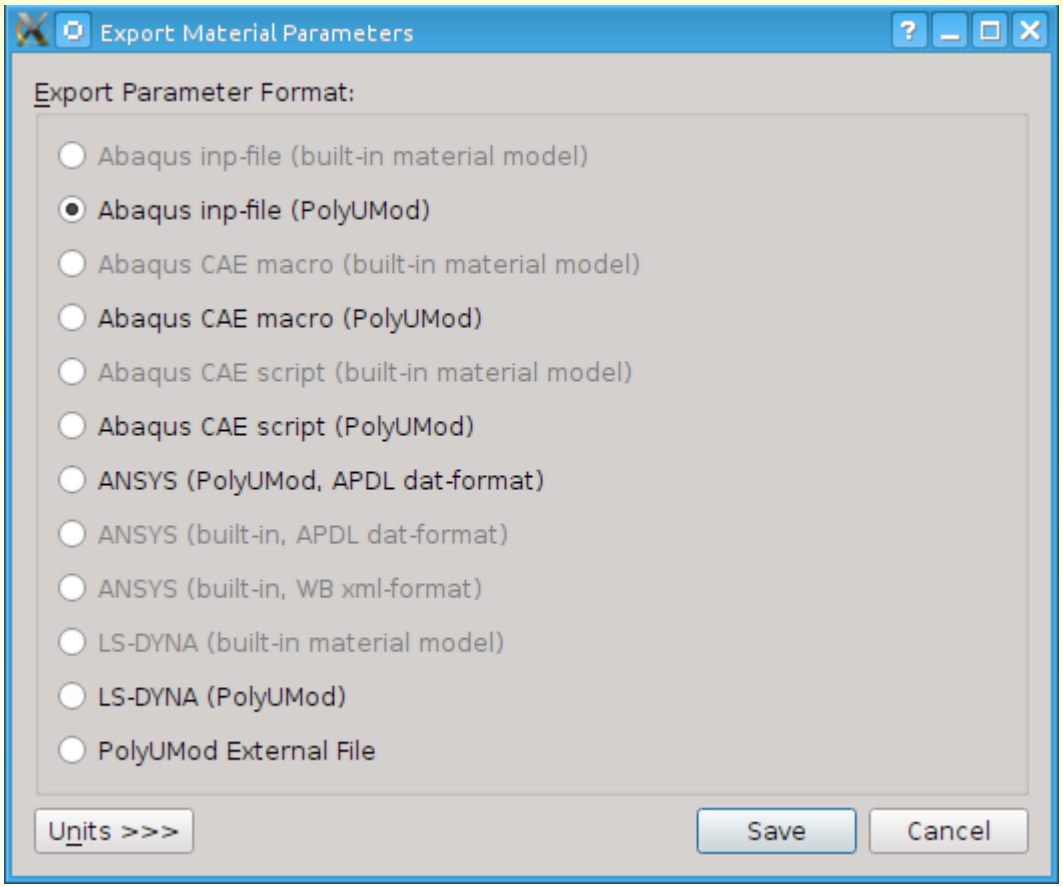
Back in the main window click the Run Once button to evaluate the new load virtual experiment.

We see that the predicted relaxes about 30% in 60 seconds.

The next step is to export the calibrated model to a FE program. Click the Export Parameters button.

Copyright Veryst Engineering , 2015

Export Material Model

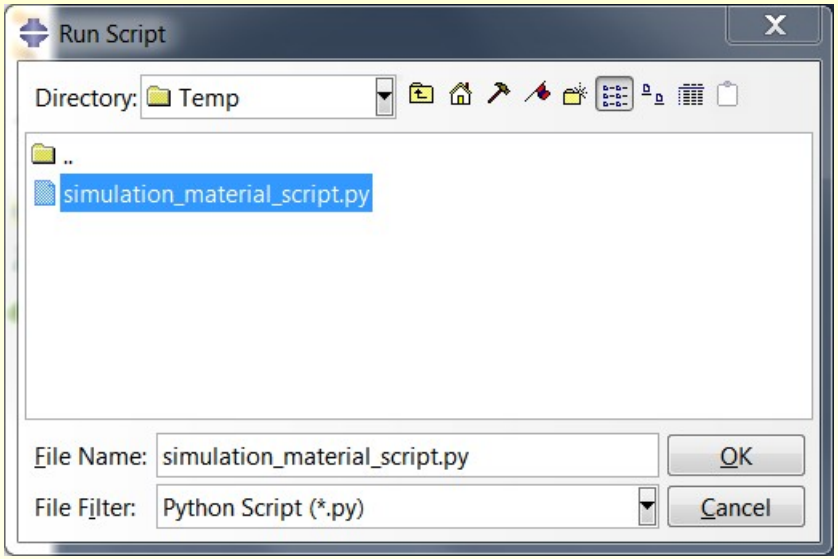


To export the material model to Abaqus/CAE select Abaqus CAE script, and click Save.

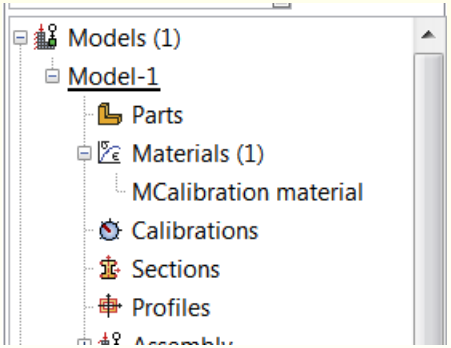
To export the material model to ANSYS select ANSYS (APDL format), and click Save.

Copyright Veryst Engineering , 2015

Import Model Into Abaqus/CAE



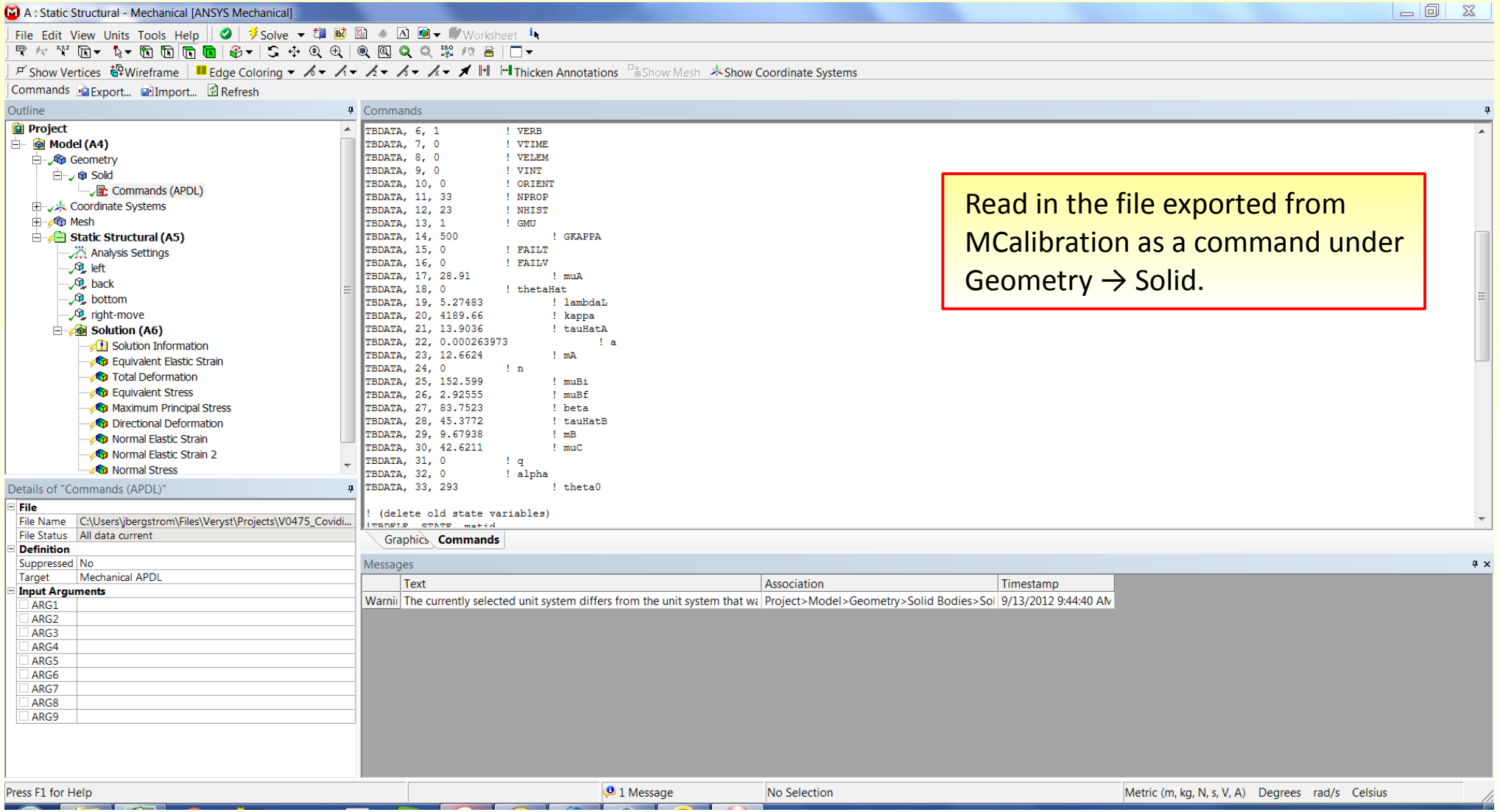
In CAE select Run Script from the File Menu, then select the script file that was created by MCalibration.



The model tree then contains the calibrated material model.

Copyright Veryst Engineering , 2015

Import Model Into ANSYS WB



Read in the file exported from MCalibration as a command under Geometry → Solid.

Copyright Veryst Engineering , 2015

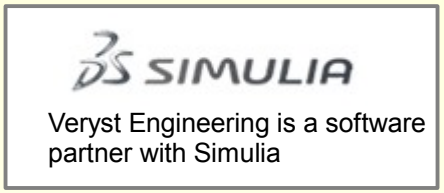
MCalibration Summary

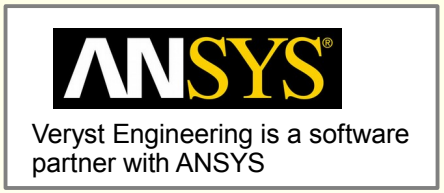
- MCalibration is an easy to use tool that can calibrate any material model.
- One of the most powerful features of MCalibration is that is can use almost any combination of experimental data, e.g. tension, compression, shear, biaxial, triaxial, stress relaxation, creep, DMA, Poisson's ratio, etc.
- MCalibration can also use direct finite element simulations of more complicated experimental tests to calibrate a material model.

Copyright Veryst Engineering , 2015

Support

- For technical support contact Dr. Jorgen Bergstrom at: jbergstrom@veryst.com





Copyright Veryst Engineering , 2015